

GerbTool











Manual

Copyright © 1989-2000 WISE Software Solutions, Inc.
All Rights Reserved






WISE Software Solutions, Inc.
2700 E. Ninth St., Suite 100
Newberg, OR 97132 USA
Phone: (503) 554-8855 ♦ Fax: (503) 554-1220





















Revised Februray 2000



















Contents




















Chapter 1: Introduction	1
Product Features	2
System Requirements.....	3
Using this Manual.....	4
If you need further assistance	4
 Chapter 2: Installation	 5
Installing GerbTool	6
 Chapter 3: Quick Start	 7
Starting Up GerbTool	8
Creating a new Design.....	8
Importing CAM Files	8
Opening an existing design.....	8
Saving design modifications	9
Exiting GerbTool	9
 Chapter 4: GerbTool Basics.....	 11
GerbTool Desktop.....	12
Menu Bar.....	13
Tool Bars.....	13
Settings Bar.....	13
 Sketch	14
 Overlay	14
 View Composites.....	14
 DRC Errors	14
 Selection Filter	14
 Grid.....	15
 Grid Snap	15
 Orthogonal Snap	15
 Arcs 360°	15
 Units and Precision.....	15






















Active Layer/D-Code/Tool	15
Coordinate Display	16
Color Bar	16
Birds Eye View	16
Split Screen Pane Dividers	16
Drawing Area	16
Cross-hair Cursor.....	17
Film Box	17
Status Bar	17
Tool Tips	17
Design Files	17
Invoking GerbTool commands.....	17
Mouse and Function key commands	17
Selecting from the main menu	18
Nested Commands	18
Interrupting a Redraw.....	20
Terminating a command	20
Dialog Boxes	20
Dialog Boxes	20
Color Chooser	20
File Chooser.....	21
Chapter 5: Performance Tips	23
Speeding up Commands	24
Nested Commands Execute Immediately.....	24
Interrupting Redraws and Highlights	24
UNDOing Edits.....	24
Programming Mouse and Function Keys	25
Memory Considerations	25
Memory Allocation Errors and Disk Space	25
Chapter 6: Example Uses of GerbTool	27
Layer Registration	28
Exporting Drill Files	28
Importing Drill Files	29
Panelizing.....	29
Viewing/Plotting 274-D Composite Layers.....	30
Drawn Pad Conversion	30
Automatic Silkscreen Clean-up	31
Creating a Soldermask	32
Transcoding	32
Snoman Pad/Trace Filleting.....	33












Chapter 7: Command Reference	35
File Menu	36
 New	36
 Open.....	36
Close	36
Merge	36
 Save	36
Save As.....	37
Import.....	37
Gerber Wizard	37
Gerber	40
Aperture List.....	41
Drill	42
Drill Tool List	42
BARCO DPF	42
HPGL	43
IPC-D-356	43
DXF	45
Mill / Rout	47
Export.....	47
Gerber	47
Aperture List.....	50
 Drill	50
Drill Tool List	51
BARCO DPF	51
IPC-D-350	51
IPC-D-356	51
DXF	52
HPGL	52
PostScript.....	54
Bitmap.....	55
Mill/Rout	56
GerbTool V8.....	56
Page Setup	57
 Print	57
Print Preview	57
Printer Setup	58
Send.....	58
Exit	58










Edit Menu	58
 Undo	59
Select	60
 New Group	60
 Add To	60
 Remove From.....	60
Invert	60
 Off.....	61
 Item.....	61
 Copy	62
 Move	62
 Delete	63
 Clip	63
 Join	63
 Rotate	64
 Mirror	64
 Scale.....	64
D-Code	64
 Transcode	65
Expand	65
 Scale.....	65
 Polarity.....	65
 Change Drill Tools	66
 Align Layers	66
Snap Pads.....	66
 Origin	67
Purge.....	67



View Menu	68
 Window.....	68
 Zoom In	68
 Zoom Out	68
 Pan	68
 All.....	69
 Film Box.....	69
 Redraw	69
 Sketch.....	69
 Overlay	69
 Grid.....	70
 Composites	70
Virtual Panel.....	70
Clear Highlights.....	70
Highlights	70
Selections.....	70
 Errors.....	71
 Selection Filter	71
Save	71
Recall	71
Previous	72
Toolbars	72
Split	72
Add.....	72
 Flash.....	72
 Draw	73
 Rectangle	73
 Vertex	73
 Drill.....	73

	Slot.....	73
	Circle	73
	Arc Ctr	73
	Arc 3 Pt.....	74
	Array.....	74
	Polygon.....	74
	Text.....	75
Setup Menu	77
	Layers	77
	Apertures	78
	Drill Tools.....	80
	Mill Tools	82
	Composites	82
	Layer Sets	83
	View/Edit	83
	Blind/Buried	84
Documentation	86
	Reports.....	86
	Apertures	86
	Drill Tools.....	87
	Mill Tools	88
Redline	89
	Add Text	90
	Add Balloon Text	90
	Add Arrow	90
	Add Line.....	90
	Sketch.....	90
	Delete	90
	Properties	91

	View Redlining	91
Drawing		91
Dimensioning		92
	Add Dimension	92
	Delete Dimension	95
	Modify Dimension	95
Properties.....		95
	Add Line.....	95
	Delete Line.....	96
	Modify Line	96
Drill		96
	Add Hole Chart	96
	Delete Hole Chart	97
	Modify Hole Chart	97
	Update Hole Charts	97
Fabrication		97
	Add Note Balloon.....	97
	Delete Note Balloon.....	98
	Modify Note Balloon.....	98
Query Menu		98
	Item.....	98
	Net.....	99
	UserData	100
Measure		101
	Point to Point	101
	Edge to Edge.....	101
	Center to Center	101
	Highlight.....	101

		Copper	102
		Extents.....	103
Options	Menu		104
		Grid Snap	104
		Ortho Line Snap	104
		Arcs 360	104
		Units and Precision.....	105
		Configure.....	105
		General	105
		Display	108
		Function Key / Mouse	110
		Ap List Converters.....	111
		Paths, Files, Extensions.....	112
		Macro Files.....	113
		User Menu.....	114
		Units and	115
		Right Click Menu.....	116
Macro		117
		Run.....	117
		Load	117
		Developer.....	117
		Record	118
Tools	Menu		118
		Panelize	119
		Automatic Panelization.....	119
		Manual Panelization	119
		Automatic Venting.....	120
		Virtual Panelization	120
		Netlist	121
		Generate.....	121
		Save	123
		DRC	124
		Snoman	130
		Teardrops	131

	Fix SilkScreen.....	132
	Pad Removal	133
	Isolated	133
	Stacked	134
	Drill	134
	Set Order	135
	Optimize.....	136
	Show Drill Path	138
	Mill / Rout	138
	Create	139
	Edit.....	140
	Properties.....	140
	Delete Plunge	141
	Reverse Direction	142
	Delete Mill	142
	Tabs	142
	Query	142
	Display	142
	Plunge/Retract	142
	Offset	142
	Sequence.....	142
	Up Path	143
	Colors.....	143
	Test Points	143
	Tools	143
	Generate	143
	Add.....	144
	Stagger	144
	Save.....	146
	Convert.....	147
	Drawn Pads	147
	Home Plate	148
	Arcs to Segmented Arcs.....	148
	Segmented Arcs to Arcs.....	149
	Gerber to Drill	150
	Drill to Gerber.....	151

To Custom	152
Raster to Vector	152
 Layer Spread	152
 Vent/Thieving.....	153
Compare Layers.....	153
Find Duplicates	154
User Menu.....	155
 Chapter 8: Aperture Conversion Rule Files	157
What is an ACR File.....	158
How does it work?	158
Regular Expression Syntax.....	158
Matching Order	160
Creating an ACR File	160
NAME	160
VERSION	161
HEADER	161
SKIP	161
DEFAULT_UNITS	162
CUSTOM	162
EXTENSION	162
DEBUG	163
XTENSION.....	163
DCODE	164
#	164
FORMAT_shape	165
FORMAT_UNITS	165
FORMAT_SPECIAL.....	166
 Chapter 9: 274-X.....	169
Embedded Apertures	170
Aperture Macros.....	170
Layer Compositing	171
Viewing Composites.....	172
Converting From 274-D To 274-X.....	172
 Chapter 10: Using Custom Apertures.....	173
Create Custom Aperture	174
 Chapter 11: Working with Text Fonts	175

Editing a Font	176
Creating a New Font	176
APPENDIX A -Aperture List File Format	177
APPENDIX B -Snoman Concepts	181
APPENDIX C -Sample Netlist File	183
APPENDIX D -Color List File	185
Glossary.....	187

Chapter 1

Introduction

Welcome to GerbTool, the easiest to use, most powerful and versatile CAM station available.

GerbTool provides a complete set of Windows based CAM tools including a feature rich and robust Drill editor for ensuring a seamless link between PCB design and manufacturing. GerbTool is designed to provide CAD/CAM professionals with the tools they need for complete control over their CAM databases. From visual verification to high level CAM tools, GerbTool simplifies and automates your PCB CAD post processing and pre-manufacturing tasks.

GerbTool's consistent and intuitive Graphical User Interface (GUI) and programmable mouse/function keys allow you to focus on accomplishing tasks quickly and efficiently, rather than on the technical details of operating the software.

Chapter 1

Product Features

- ◆ Fast and very easy to use, less user fatigue.
- ◆ Accurate to 1/100 micron.
- ◆ Complete UNDO to beginning of session.
- ◆ Full Design Rule Checking (DRC) including Annular Ring Checking and Stub Detection.
- ◆ Automatic DRC corrections including Pad Shaving.
- ◆ SNOMAN™ and Teardrop Pad/Trace filleting.
- ◆ Fully automatic Panelization and Venting.
- ◆ Advanced Drill Optimizing including Step & Repeat.
- ◆ Milling/Routing/Profiling including Tabs.
- ◆ Test Point/Net List extraction with Automatic Staggering and Multiple Fixture Plate support.
- ◆ IPC-D-356 Import/Export.
- ◆ Fabrication Drawing Tools including Auto Dimensioning, Hole Charts, Note Generation and Construction Lines.
- ◆ Track changes with Compare layers tool.
- ◆ Isolated, Redundant and Buried Pad Removal.
- ◆ Automatic removal of Silkscreen data from pads.
- ◆ Full support for true multi-layer netlists including net highlighting.
- ◆ Easy to use Redline/Markup Documentation tools.
- ◆ Scalable check plots to HPGL, PostScript®, Laser printers and all printers/plotters supported by MS Windows.
- ◆ Conversion of drawn pads to flashes.
- ◆ Macro Language allows the addition of new commands.
- ◆ Photo plotter support includes 274-D, 274-X, FIRE9XXX, EIE, BARCO DPF and IPC-D-350.
- ◆ Accurate display of pwr & gnd plane composites.
- ◆ Aperture scaling to create soldermasks, shrink/expand traces etc.
- ◆ Ability to scale layer(s) to shrink or expand the database.
- ◆ Merge a complete design into another.
- ◆ Import DXF / Drill / Mill-Rout / HPGL / BARCO files.
- ◆ View up to 999 layers simultaneously.
- ◆ Handles 9999 apertures.
- ◆ Aperture list conversion tools allow the addition of custom aperture list converters.
- ◆ Easily created custom apertures and custom fonts.

System Requirements

The following section describes the *minimum* system configuration required.

Note: The following configuration should be considered absolute minimums. Your actual system configuration may need to be enhanced depending on the size of your Gerber databases and your desired level of performance.

- ◆ IBM PC or compatible with at least a 200Mhz Pentium CPU and Windows 95, 98 or NT.
- ◆ At least 16 meg of RAM.
- ◆ At least 30 megs of free disk.

Chapter 1

Using this Manual

This manual was designed to assist the CAD/CAM professional in utilizing GerbTool's features as efficiently and as quickly as possible. Chapter 3, 'Quick Start', is especially geared toward providing the information you need to become productive immediately. A prior knowledge of CAD/CAM concepts and your computer's operating system is assumed.

If you need further assistance

If you purchased GerbTool from an authorized reseller, you should contact them directly for technical support as they will have a better understanding of your needs and intended uses of the software.

If you are unable to get satisfactory assistance in solving your problems, you may check our web site at www.gerbtool.com for uptodate FAQ's, send email to tech@wssj.com, or send a FAX message to WISE Software Solutions, Inc. at (503) 554-1220. If you do not have access to email or a FAX machine, you may call (503) 554-8855 between the hours of 8:30 am and 3:00 pm, Pacific time.

Chapter 2

Installation

This Chapter describes the basic installation of GerbTool. GerbTool provides a Setup program that checks your system and asks a series of questions on how you want to install the product. The installation process also decompresses the files on the program disks so they can be opened and used on your hard disk.

Chapter 2

Installing GerbTool

This section describes the process of installing the GerbTool software and configuring the license manager.

Insert the GerbTool CD-ROM in the appropriate drive and double click Setup or click Start, Run and then type:

```
<drive>:\setup
```

Follow the installation instructions displayed on the screen. To obtain the required access code, run the GerbTool License Manager by clicking on the following icon:



Simply fill in the registration information section and then click on the Print button. This will print a 'License Request Form' that should be faxed directly to the fax number shown on the document. A valid license will be faxed back to you usually within a few hours.

Be sure to read the 'readme.lst' file for features that have been added, or changes that may have occurred, since this manual was printed.

Chapter 3

Quick Start

To get started quickly, this chapter provides a quick overview in using GerbTool. A more comprehensive description for each individual GerbTool function is provided in Chapters 4 - 8.

Chapter 3

Starting Up GerbTool

To start GerbTool double click on the GerbTool icon:



Creating a new Design

To create a new design, select the File/New command. This will clear your workspace and allow you to begin a new and initially untitled design. You may then begin adding data manually or importing files into your design.

When you are ready to save your new design for the first time, GerbTool will prompt you to choose a new filename for your design.

Importing CAM Files

Before you can begin working on a new design you typically import one or more CAM files into GerbTool. While the most common types of files imported into GerbTool are Drill and Gerber, the File/Import menu contains commands for importing other essential CAM file formats. In addition, when importing Gerber files, external aperture list files are often provided. GerbTool provides aperture list conversion for most of the popular CAD and photo-plotter aperture list formats in use today. This conversion process will translate a CAD aperture list directly into GerbTool reducing data entry related problems.

Note: Aperture list files are not required for 274-X or FIRE9000 format Gerber files as they are embedded in the Gerber file.

Opening an existing design

To open an existing design, select the File/Open command.

Saving design modifications

Select the File/Save or File/Save As command to save your current design.

Note: This command saves the entire contents of your current design in a single GerbTool design file. To create Gerber and other CAM files, use the File/Export commands.

Exiting GerbTool

To exit GerbTool select the File/Exit command. If any layers have been modified, GerbTool will request confirmation that you really want to exit.

Chapter 4

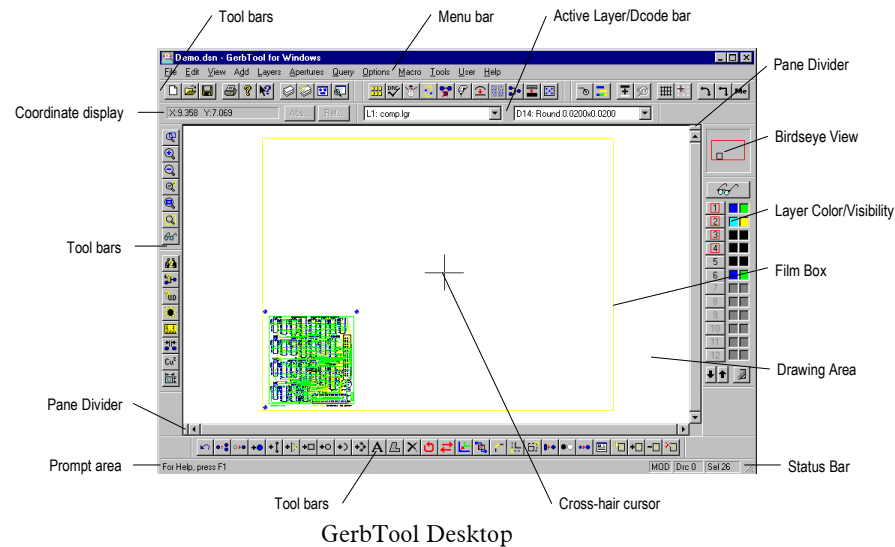
Chapter 4

GerbTool Basics

This Chapter provides information on the fundamentals of operating GerbTool, i.e. 'The Basics'.

Chapter 4

GerbTool Desktop



The GerbTool desktop consists of the following components:

- ◆ Menu Bar where you can ‘pull down’ command menus.
- ◆ Docked or floating Tool bars where you can invoke commands with a single click.
- ◆ Docked or floating Settings control bar where you control various program settings such as Metric display mode and Grid Snap.
- ◆ Docked or floating Active Layer/D-Code/Tool control bar where you may specify the currently active layer, D-Code or tool for editing commands.
- ◆ Docked or floating Coordinates control bar area where the current Cross-hair X-Y coordinates are displayed.
- ◆ Docked or floating Color dialog bar that allows quick changing of layer colors and visibility.

- ◆ Docked or floating Birdseye view that shows the current view window relative the extents of the loaded database.
- ◆ Adjustable pane dividers to split the drawing area into multiple views.
- ◆ Drawing Area(s) where all database items are displayed.
- ◆ Cross-hair cursor indicating the position of the mouse within the drawing area.
- ◆ Film Box graphic that indicates the size of the current film box.
- ◆ Status bar with prompt area where GerbTool commands issue user prompts and display command status.
- ◆ Tool Tips on most desktop features including toolbar buttons, control bar buttons and menu items.

Menu Bar

The GerbTool menu bar appears across the top of the desktop display. Each word in the menu bar represents a menu of related commands. When you select a word in the menu bar by moving your mouse over the word and clicking the menu button, that menu will ‘pull down’. Each item on the pulled down menu may be executed by selecting it.

Tool Bars

GerbTool tool bars may be docked to an edge of the desktop or may float in a small window. Each icon within the tool bar represents a shortcut method of invoking a command. When you click on a icon in the tool bar, the command associated with that icon will be invoked.

Note: All tool bars and control bars may be moved to a location you prefer by clicking your mouse on the bar and dragging the bar to a new location. They may be attached to an edge of the desktop (docked), or float in a small dialog box.

Settings Bar

The Settings control bar allows you to quickly and easily control various program options with a single mouse click. This section describes each button within the Settings control bar.

Chapter 4



Sketch

This button toggles sketch mode on/off. When sketch mode is enabled, items are shown with an outline only. An option to this setting is stick mode. If enabled, stick mode displays all draws as a single thin line. Stick mode is enabled using the Options/Configure/Display page. This mode can help you spot stacked and buried items.



Overlay

This button toggles overlay mode on/off. When overlay mode is enabled, items become transparent when drawn atop each other. When overlay mode is disabled, items obscure whatever is drawn previously. Overlay mode makes it easier to spot stacked and buried items. Flashes also display significantly faster in overlay mode.



View Composites

This button toggles the way composite layers (274-X and FIRExxxx only) are displayed. When this button is enabled the polarity of each layer, specified using the Setup/Composites dialog, will be honored. If a layer is specified 'Clear', all data in that layer will be displayed with the background color.

Note: When View Composites is enabled, the active layer is not displayed on top as it normally is.



DRC Errors

This button toggles the display of rule violation errors on/off. If DRC errors exist and this setting is enabled, GerbTool will display the DRC View Errors dialog box. See Chapter 7, 'DRC' for more information on this topic.



Selection Filter

This button toggles the display of the current 'Selection Filter' if any. Many editing commands present a selection filter dialog which you may toggle on/off using this button or the nested command 'F'.



Grid

This button toggles the system grid display on or off. See Chapter 7, 'Options' for more information on grids.



Grid Snap

This button toggles grid snap mode on/off. When grid snap mode is enabled, your cross-hair cursor will automatically jump to the nearest grid point. See Chapter 7, 'Options' for more information on grids.



Orthogonal Snap

This button allows you to toggle orthogonal snap mode on/off. When enabled, all lines drawn interactively will be forced to the specified angle.

Note: The current setting may be temporarily overridden by holding down the Ctrl key.



Arcs 360°

This button toggles the method of creating arcs used by the Add/Arc and Add/Circle commands. If enabled all arcs will be created using 360° circular interpolation. If disabled, all arcs will be created using small line segments. This does NOT affect the way Gerber data is read from a disk file. It only pertains to adding new arcs with the Add/Arc commands.



Units and Precision

This button serves as a shortcut to the Options/Configure/Units and Precision dialog. See Chapter 7 for more information.

Active Layer/D-Code/Tool

The Active Layer/D-Code/Tool control bar may be docked to the top or bottom of the desktop or float in a small dialog box. This control bar allows you to control the currently active layer and d-code or tool

Note: You may also use the nested commands L/<ctrl>L or double click a layer visibility button on the Color Bar to change the active layer. D/<ctrl>D may be used to change the current D-Code.

Coordinate Display

The coordinate display dialog bar may be docked to the sides of the desktop or float in a small dialog box. It shows you at a glance the current location of the Cross-hair cursor. The format of the display is controlled by the current setting of the Options/Configure/Units and Precision dialog.

Color Bar

The layer color dialog bar may be docked to the sides of the desktop or float in a small dialog box. It is available at all times to change layer colors and visibility. When a layer is on, indicated by a red box around the layer number, it is both visible and editable. When a layer is off it is neither visible nor editable. When a layer is ref, indicated by a black box around the layer number, it is visible but not editable. You may stretch the dialog vertically to show more or less layers.

Note: Double clicking a layer visibility button makes the layer specified in the button the active layer.

Birds Eye View

The Birds Eye view may be docked or float in a small dialog box. The black rectangle represents the database extents while the red rectangle represents the current viewing window. This allows you to tell at a glance exactly where your current view window is located.

Split Screen Pane Dividers

By clicking and dragging the adjustable pane dividers you may split the drawing area into up to four separate viewing windows. Each window represents a different view of your design. This allows you to view and edit your data at multiple zoom levels or locations simultaneously.

Drawing Area

This is the area between the Menu Bar and the Status Bar. All database items are displayed here.

Cross-hair Cursor

While the mouse position is within the Drawing Area, the cursor will be displayed as a cross-hair. The exact location of the cross-hair cursor is displayed in the Coordinate Display dialog bar described above.

Film Box

The film box represents the size of the film that you will plot on. The film box is a graphic display only. It does NOT become part of your Gerber database(s). You may control the size and color of the film box with the Options/Configure command detailed in Chapter 7, 'Options'.

Status Bar

GerbTool displays command status and prompts in this area.

Tool Tips

By holding your mouse cursor over a desktop feature for a second or two, a small popup window will appear with a short description of the feature.

Design Files

GerbTool saves all layer data and job specific data in a design file. This file is a simple human readable ASCII file. The format specification for an GerbTool design file is available upon request.

Invoking GerbTool commands

This sections describes the different ways you may invoke GerbTool commands.

Mouse and Function key commands

GerbTool comes pre-configured with the following mouse and function key assignments:

Chapter 4

Key	Assignment
LBUTTON	ViewWindow
MBUTTON	ViewZoomIn
RBUTTON	None
F1	Help
F2	ViewFilmBox
F3	ViewPrevious
F4	ViewAll
F5	SetupLayers
F6	SetupApertures
F7	DocReportApertures
F8	QueryHighlight
F9	QueryItem
F10	Menu
F11	EditSelectAdd
F12	EditSelectRemove

Pre-configured Mouse/Function key assignments

The assigned mouse and function key commands are available anytime GerbTool is idle, i.e. there is no command prompt in the prompt area.

Note: For complete information on customizing your mouse and function keys, refer to the Options/Configure command in Chapter 7.

Selecting from the main menu

At anytime you may position your cursor in the main menu bar and select a command by clicking a mouse button. If you complete a selection, any previous command will be terminated before executing the new selection.

Nested Commands

Nested commands are available anytime GerbTool has prompted you to enter a point or is idle. See below for a list of the available nested commands:

Key	Command
Enter	Enter coordinate at cursor location
Home	Snap cursor to center of item
PgUp or Num+	View/Zoom In
PgDn or Num-	View/Zoom Out
Left Arrow	Scroll page left
Right Arrow	Scroll page right
Up Arrow	Scroll page up
Down Arrow	Scroll page down
I	View/Zoom In
O	View/Zoom Out
A	Turn on all layers
Ctrl+A	Turn off all but active layer
C	Enter absolute coordinates
Ctrl+C	Enter relative coordinates
D	Increment current D-Code/Tool
Ctrl+D	Decrement current D-Code/Tool
F	Popup Selection Filter Dialog
Ctrl+F	Edit configuration flags
Ctrl+G	Edit system grid
H	Toggle highlights on/off
Ctrl+I	Screen Print
L	Increment active layer
Ctrl+L	Decrement active layer
Shift+L	Setup Layer Sets
M	Run Macro
Ctrl+M	Change Units & Precision
P	View/Pan
Ctrl+P	Toggle Auto Pan mode
Ctrl+Alt+Q	Quit immediately without confirmation
R	View/Redraw
Ctrl+R	View/All
S	Toggle grid snap
Ctrl+S	File/Save
T	Toggle Mill/Rout display
U	UNDO last edit
Ctrl+U	UNDO All edits
V	Toggle composite viewing
Ctrl+V	Toggle Virtual panel mode
1-9,0	Bring a layer to the top (1-10)
Ctrl+1-9,0	Bring a layer to the top (11-20)

Nested Commands available.

The above commands will be executed immediately without effecting the current command.

Chapter 4

Interrupting a Redraw

Anytime GerbTool is redrawing the display or highlighting a window of data you may halt the drawing process by touching the [Esc] key or clicking the right mouse button. This will not affect the operation of the command and in many cases will speed up the operation of a command.

Terminating a command

You may terminate a command, or at least one level of a multi-step command, by touching the escape key or right mouse button.

Dialog Boxes

GerbTool makes use of dialog boxes and the standard Windows Color Chooser and File Chooser to obtain information from, and communicate with, you; the user.

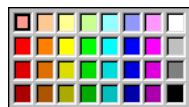
Dialog Boxes

Dialog boxes are used to enter information into GerbTool. They may contain data entry fields, checkable buttons, color buttons, help buttons, scroll bars and exit buttons (OK, Cancel, Close).

Note: Most GerbTool dialog boxes have a help button with a question mark located in the title bar. To use this feature, click the help button and then click on a control within the dialog box. A popup window will appear with help specific to that control and dialog box.

Color Chooser

GerbTool provides many opportunities for you to control the color of objects on the GerbTool desktop. In all cases GerbTool presents a dialog of 32 color buttons to choose from.



Color dialog.

To dismiss the dialog without choosing a color, touch the *Esc* key. To select a color, left click the desired color. To change the color choice within the dialog button, right click the desired color. This will popup the standard Windows color chooser.

File Chooser

The file chooser provides a convenient way of selecting filenames. There are two forms of the file chooser form. The first, allows you to select only one filename. You select the desired file by clicking directly on a filename. The chosen filename appears in the Filename field.



Example File Chooser Form.

The second, allows you to select multiple filenames by holding down the control key while clicking your mouse directly over each desired filename. Each selection remains highlighted. The behavior of the file chooser depends on the dialog/field that you are currently editing.

Chapter 5

Chapter 5

Performance Tips

This Chapter provides tips on obtaining optimal performance from GerbTool.

Speeding up Commands

The following two sections describe short cuts that are available to speed up most GerbTool operations.

Nested Commands Execute Immediately

A very powerful feature of GerbTool is the availability of nested commands. These commands are available at all times when GerbTool is waiting for you to enter a coordinate (point) or is idle (i.e. no command has been selected). With these commands you can move around, snap to the center of a database item, change what layers are viewed, undo edits, and so on. Please refer to Chapter 4, 'Nested Commands' for a complete list of available nested commands.

Interrupting Redraws and Highlights

Any command that redraws the database or highlights a group of items can be sped up by canceling the drawing process. By clicking the right button or touching the escape key, you can halt whatever is redrawing the display. This doesn't affect the operation of the command, only the redraw is effected. Once you're comfortable with the operation of GerbTool commands you will find that this ability significantly speeds things up.

UNDOing Edits

The Undo command provides a high level of freedom when making database edits. You may experiment and try 'daring' edits without fear of data loss when undo is enabled. Since undo is available as a nested command 'U' you may undo edits immediately without even having to exit the current command! Undo works for all edits regardless of size and there is no limit to the number of edits you can undo. Just remember to enable undo with the Options/Configure command **before** making your edits. Then use the Edit/Undo or the nested command 'U' to undo as desired.

Programming Mouse and Function Keys

GerbTool's easy to use GUI (Graphical User Interface) is further enhanced with the versatility of programmable mouse and function keys. Using the Options/Configure command you may program the mouse buttons and the function keys F1-F12 with commands that **you** frequently use and in a layout that **you** find comfortable. Refer to Chapter 7, 'Options' for more information on how to program your mouse and function keys.

Memory Considerations

GerbTool was developed to operate in a true 32-bit environment with virtual memory. This allows GerbTool to address the entire memory range of the CPU even if the actual installed amount of RAM memory is less (i.e. 16 mb).

Note: While virtual memory is a very powerful feature, there is no substitute for RAM memory for maximum speed. For example, if you load 20 mb of Gerber files into GerbTool on a 16 mb system, you will notice a lot of disk activity as the virtual memory manager begins to 'thrash' due to the disproportionately small amount of real memory.

Memory Allocation Errors and Disk Space

If you receive an error message such as 'memory allocation error', this indicates that your system has exhausted its allocated swap space. You can help keep the swap file usage down by occasional use of the Edit/Purge command (see Chapter 7, 'Purge') and by disabling the undo feature if not required. Purging compacts GerbTool's internal database and allows more efficient use of memory.

Chapter 6

Chapter 6

Example Uses of GerbTool

This Chapter provides several examples of the kinds of tasks that can be accomplished quickly and easily with GerbTool.

Layer Registration

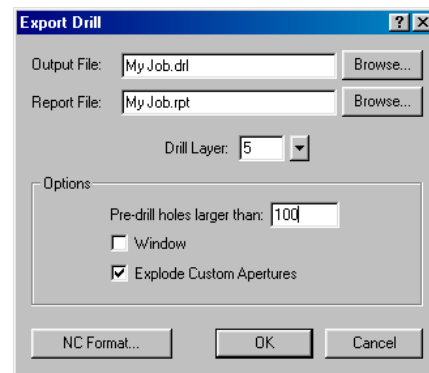
Layer registration involves lining up all layers so that when multiple layers are viewed simultaneously they appear correctly aligned. Proper layer alignment is also crucial to the successful creation of a multi-layer netlist.

Steps:

- Determine the layer to which all other layers should be aligned with; a master layer.
- Choose an item on the master layer to use as a reference point.
- Invoke the Edit/Align command and select the item you chose as a reference point.
- You then select an item, on each layer to be aligned, that corresponds to the reference point. As you select each additional item, the entire layer will be automatically aligned with the master layer.

Note: Remember, you may use the nested zoom in/out and pan keys (see Chapter 4, ‘Nested Commands’) to make it easier to home in on the reference and corresponding items.

Exporting Drill Files



Export Drill dialog.

Using the File/Export/Drill command, GerbTool allows you to create an NC Drill file. The format of the created drill file is selected by clicking on the NC Format button within the Drill dialog box as shown above.

Note: Use the Documentation/Reports/Drill Tools command to determine if all drill locations have a proper tool assigned.

Panelization of the image should be performed prior to executing this command. If your drilling equipment has a small memory capacity you should perform a 'virtual' panelization. This will allow GerbTool to insert the needed step & repeat codes into the output drill file. Preferably, if your drilling equipment has enough memory, you should perform a normal non-virtual panelization. This will result in a fully optimized panel for the maximum in efficient drilling.

Importing Drill Files

Using the File/Import/Drill command, you may load a NC Drill file into the active drill layer.

When importing a Drill file, GerbTool may prompt for drill tool sizes if they're are not specified within the input file. If tool sizes are specified, new tools will be created as needed. Use the Setup/Drill Tools to change drill tool sizes.

GerbTool converts step & repeat patterns into Custom apertures. GerbTool will subsequently output all repeated Custom apertures as step & repeat when exporting drill files.

Note: Use the Documentation/Reports/Drill Tools command to determine what tools are currently in use.

Panelizing

GerbTool makes panelizing a very simple 'One Step' process when using the Auto Panel feature. After turning on only the layers to be panelized and selecting the Tools/Panelize command, simply ensure the Auto Panel button is checked, as shown in Chapter 7, 'Tools/Panelize', and enter the minimum image border to border spacing in the X and Y fields. The spacing specified should be between adjoining edges of the intended images. GerbTool will automatically calculate the maximum number of images that will fit inside the current film box. After asking for confirmation, GerbTool will complete the panelization process. Depending on the setting of the Virtual button, GerbTool will either

Chapter 6

copy the proper number of images into the database or simply note the number of copies and their location for display purposes.

Note: You may right click or touch the escape key to stop the drawing process anytime during the panelizing process. This usually provides a noticeable improvement in the overall time to complete the panelizing process without effecting the finished panel in any way.

Viewing/Plotting 274-D Composite Layers

By allowing the use of black and white for layer colors, GerbTool allows accurate viewing of composite pwr & gnd layers. Setting the negative layer to white on a black background and the positive layers to black will result in a realistic depiction of the final film.

Note: Since the negative layer must be displayed first, it is important that the negative layer be before the positive layers (i.e. a lower layer number) and not the active layer.

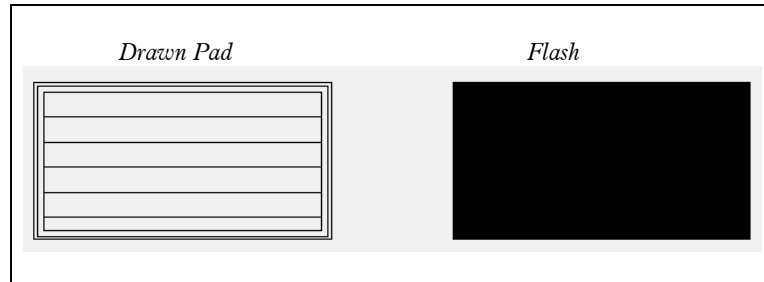
Plotting a composite layer on a printer is equally simple. Just view your composite layers as described above and then use the File/Print command. The plotted image will appear on the page exactly as it does on the display.

Note: Since the image for printing is created in a high resolution off screen bitmap, the film box and display grid may appear on the output page. You can disable this by setting the film box color to the background color using the Options/Configure command and disabling the display of the grid using the Options/Configure, or nested 'G', command.

Drawn Pad Conversion

Unfortunately, many CAD systems still output rectangular pads, such as surface mount pads, using multiple draws to 'fill in' the rectangle rather than a more efficient single flash. This results in bloated Gerber files and increased processing times. Also, it is virtually impossible for high level CAM tools such as DRC to recognize these drawn pads as a pad rather than just a bunch of traces. Shown below are the difference between a typical drawn pad and a comparable flash.

Example Uses of GerbTool



Drawn pad versus a flash.

The drawn pad shown requires 27 separate Gerber commands to accomplish what one Gerber flash can accomplish. As you can see, if you have 2000 of these drawn flashes, you'll have a Gerber file with at least 54,000 lines when flashes could accomplish the same thing in only 2000!

Fortunately, GerbTool provides a powerful tool to convert these drawn pads to flashes. Using the Tools/Convert/Pads command, you may simply and quickly convert all your drawn pads to flashes. You do this by identifying just one occurrence of a drawn pad and allowing GerbTool to find all drawn pads that match. And to increase GerbTools ability to recognize matching drawn pads, you can specify a tolerance value to compensate for some CAD systems round off errors. By specifying a tolerance you allow GerbTool to relax its criteria for determining matching drawn pads.

Note: It is recommended that the conversion of drawn pads to flashes be the first thing done to your designs. This will usually ensure complete and trouble free conversion. Also, you must convert all drawn pads to flashes **before** generating a netlist or running most other CAM tools.

Automatic Silkscreen Clean-up

A very powerful feature of GerbTool is its ability to automatically clean-up a silkscreen where lines touch or are too close to the pads. Using the Tools/Fix SilkScreen command you specify the layer(s) that the silkscreen and pad master are on and the minimum spacing that must be maintained between the silkscreen data and the pads. If desired, you can use window mode to clean-up isolated areas rather than the entire silkscreen layer. GerbTool will then 'clean-up' all places where silkscreen lines are too close to a pad. Each offending line is moved just enough to eliminate the violation and no more. See Chapter 7,

Chapter 6

‘Tools/Fix SilkScreen’ for more information and before and after illustrations.

Creating a Soldermask

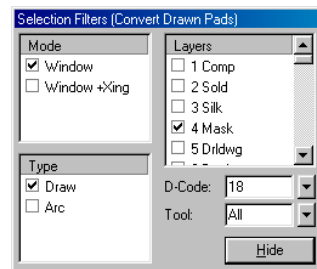
Creating a soldermask is a simple and easy process using the Edit/D-Code/Scale command.

Steps:

- First create the soldermask layer by using the Edit/Copy command to copy the pad master layer onto an empty layer.
- Use the Setup/Layers command to change the layer type of the new layer to one of “Top Mask” or “Bot Mask”.
- Now, select the Edit/D-Code/Scale command, enter a scale factor for both X & Y and click on the Fixed Amount check box. Click on the OK button. GerbTool will add apertures to the aperture list as necessary and replace the D-Codes with the new scaled D-Codes. The original D-Codes within the aperture list are not modified.

Transcoding

Using the Edit/D-Code/Transcode command, you may transcode (“transform D-Code”) either item by item or by selecting a group. Using the ‘Selection Filter’ you may be very selective, if desired, as to which D-Codes are transcoded. For example, to transcode only draws with a D-Code of D18 only on layer 4 and only within a particular window, the following selection filter would be required:



Restrictive Selection Criteria.

After selecting and highlighting the desired D-Codes GerbTool will prompt you for the new D-Code and then perform the actual transcoding.

Snoman Pad/Trace Filleting

Snoman is a highly configurable form of the method of eliminating pad/trace separation that is often referred to as filleting or teardropping (see APPENDIX B for a technical description of Snoman). The purpose of Snoman is to increase your manufacturing yield by adding more copper in the area of the pad/trace junction thereby eliminating any possible pad/trace separation. Snoman is used primarily when dealing with very small pads and traces such as micro vias in the 30 mil or less range but can be used anywhere to prevent pad/trace separation. Snoman provides additional versatility by allowing user control of the size and location of the generated Snoman pads along with an integral DRC to eliminate any possible spacing violations. Please refer to Chapter 7, 'Snoman' for a complete description of using the Snoman tool.

Trivia: Snoman derives its unusual name from the appearance of a Snoman pad placed on-top of a host pad which resembles a real 'snowman'.

Chapter 6

Chapter 7

Command Reference

This Chapter provides details of invoking and using each GerbTool command.

File Menu

This main menu selection presents a menu of commands for dealing primarily with files and printing. The menu selections include:

- | | |
|-----------|-----------------|
| ▪ New | ▪ Export |
| ▪ Open | ▪ Page Setup |
| ▪ Close | ▪ Print |
| ▪ Merge | ▪ Print Preview |
| ▪ Save | ▪ Printer Setup |
| ▪ Save As | ▪ Send |
| ▪ Import | ▪ Exit |



New

The New command clears the current workspace and allows you to begin a new design.



Open

This menu selection allows you to open an existing GerbTool design.

Close

This menu selection closes the current design and disables all GerbTool functions except File/New and File/Open. This command is not normally used and is retained for primarily historical reasons.

Merge

Selecting this command allows another complete design to be merged layer by layer into the current design.



Save

Select this menu item to save the current design. This command does not clear the current design; you may continue to work on the current design after saving.

Save As

Select this menu item to save the current design under a different filename. This command does not clear the current design; you may continue to work on the newly named design after saving.

Import

The Import menu item presents a sub-menu with the following commands:

- | | |
|-------------------|---------------|
| ▪ Gerber Wizard | ▪ BARCO DPF |
| ▪ Gerber | ▪ HPGL |
| ▪ Aperture List | ▪ IPC-D-356 |
| ▪ Drill | ▪ DXF |
| ▪ Drill Tool List | ▪ Mill / Rout |

Gerber Wizard

This command presents the 'Gerber Import Wizard'. This wizard makes importing Gerber files, Drill files and aperture lists extremely easy. The wizard allows you to specify which data files and aperture lists you wish to import, assign aperture lists to specific Gerber files and order the loading sequence. You may import files using any combination of Gerber/Drill dialects and aperture list formats. This command begins importing the specified files into the first empty block of consecutive layers found. Multiply defined Dcodes/Tools are automatically remapped to new Dcodes/Tools as needed.

Page 1.

This page allows you to select the folder that contains the files to import. The following configuration options are also provided to help you fine tune the wizard for the fastest automatic file detection.

Configure file types to ignore

Pops up the program configuration dialog where you may enter any filename extensions that you know are not used for Gerber, Drill or aperture list files. The more files that may be safely ignored, the faster the Gerber Import Wizard will be able to scan the specified folder.

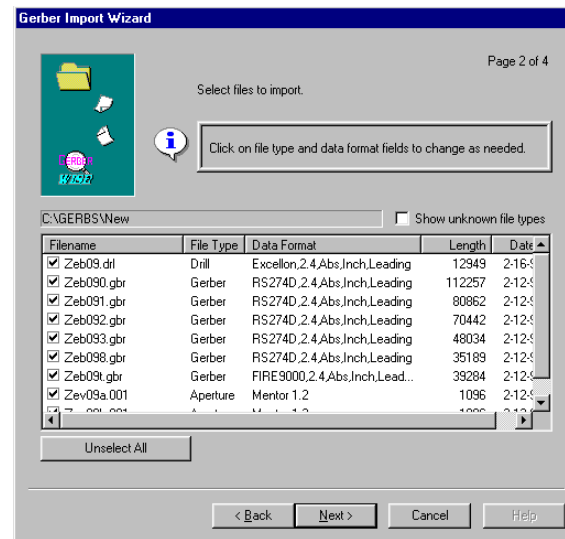
Configure aperture list converters

Click this button to configure what aperture list converters should be enabled. The more aperture list converters enabled, the longer it will take to scan the specified folder if there are files not recognized by the wizard.

Chapter 7

Page 2.

This page presents the list of Gerber/Drill files and apertures lists that GerbTool found in the import folder.



Gerber Import Wizard page 2.

Clicking on the list column headers will sort the list based on the data within the respective columns.

Filename

Indicate whether GerbTool should import a file by putting a check mark by the filename.

File Type

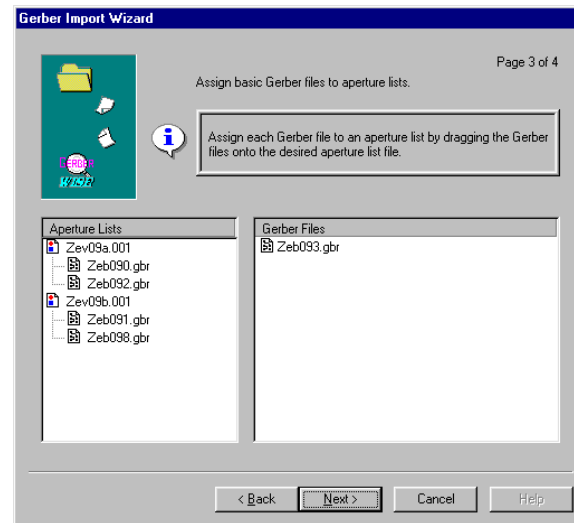
If GerbTool detects the file type incorrectly, you may change it by clicking on the appropriate file type. A pop up menu will appear where you may choose Gerber or Aperture.

Data Format

This column indicates the detected data format for each file. To override the detected format, click on the appropriate entry. The Gerber Import data format dialog or Aperture list format selector will pop up where you may make any necessary changes. The Gerber Import data format dialog is explained in the following File/Import/Gerber command section.

Page 3.

Use this dialog to assign basic Gerber files to aperture lists. Drag and drop each Gerber file onto the desired aperture list filename. This dialog will not list any extended Gerber files as their aperture lists are embedded in the Gerber file.



Gerber Import Wizard page 3.

Page 4.

This dialog presents the final list of Gerber files that are to be imported and allows you to order them as desired. Click, drag and drop each list item to the desired position.

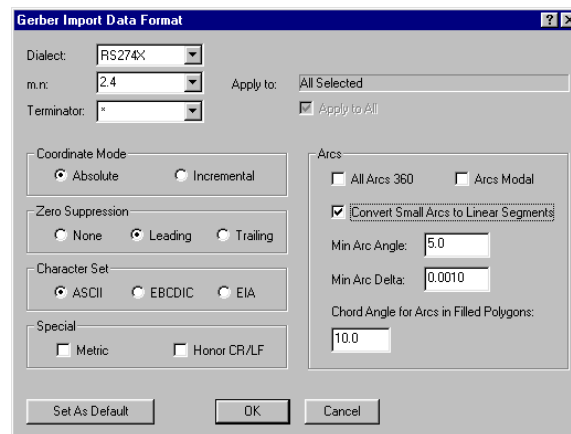
When you are satisfied with your selections and ordering, click the Finish button. Your files will be imported and a log file will be generated that lists the status of the import process. Whether you choose to suppress warning messages or not, the log file will be saved as 'ImportGerber.rpt' in the current design folder.

Chapter 7

Gerber

This command allows you to import one or more Gerber files into the current design. All files selected must be of the same file format. This command begins importing the specified files into the first empty block of consecutive layers found. If the files being imported are basic Gerber files, an Aperture list must be imported prior. See the Import/Aperture List command below for specific information on how previously defined Dcodes are remapped and assigned to subsequently loaded Gerber files. Extended Gerber files contain their own aperture information and Dcodes that are already defined are automatically remapped to new Dcodes as needed.

Before importing the selected files, GerbTool allows you to override the detected file format with the following dialog:



Gerber Import data format dialog.

Dialect

Indicates the specific dialect of the Gerber language such as RS274D, RS274X and FIRE9XXX. If in doubt, choose RS274D.

m.n

Coordinate Format such as 2.3. This specifies 2 decimal digits before an implied decimal point and 3 following. (i.e. 12250 represents 12.250 if the coordinate format is 2.3).

Terminator

Indicate the block terminator (EOB) as either '*' or '\$'.

Coordinate Mode

Choose Absolute or Incremental (See Glossary for descriptions of these terms).

Zero Suppression

Indicate whether leading zeros are suppressed, trailing zeros are suppressed, or no zero suppression.

Character Set

Indicate the expected character set.

Special

You may enable Metric mode indicating that your files are in metric format and if carriage return and line feeds should be honored as block terminators.

Arcs

Specify whether all circular interpolated arcs should be considered 360°, whether arcs should be considered modal, whether to convert small arcs to linear line segments and the chord angle used in segmenting arcs found in filled polygons. If small arcs are to be converted to linear line segments, the two parameters that affect this conversion are:

Min Arc Angle

This specifies the smallest sweep angle that a arc may be before it is selected for linear conversion.

Min Arc Delta

This specifies the smallest distance between end points of an arc before it is selected for linear conversion.

The critical format items are 'm.n', coordinate mode and whether trailing zeros are suppressed. If a design is loaded with an incorrect format specified, GerbTool will attempt to display it with unpredictable results. If you inadvertently import a Gerber file this way, Undo and re-import.

Aperture List

This command allows you to import an Aperture List file into GerbTool. Specify the file to convert and choose the aperture list format using the pull down list. If you are not sure which format to choose, select the 'Auto Detect' option and let GerbTool determine what the format is automatically.

Note: See ‘Options/Configure’ later in this chapter for information about adding additional aperture list converters to GerbTool. See Chapter 8, ‘Aperture Conversion Rule Files’, for information on creating your own aperture list converters.

All Dcodes that were previously defined will be remapped to new Dcodes and an internal remapping table will be created. Therefore, any Gerber files subsequently imported with the above Import/Gerber command will have their Dcodes remapped accordingly. This procedure of importing an Aperture list followed by one or more basic Gerber files may be repeated as many times as needed as long as there is room for more apertures. The Import/Gerber Wizard makes easy work of importing multiple sets of Gerber files, each with their own Aperture list, in one simple session.

Drill

This command allows you to import a NC Drill file into the currently active drill layer. Drill tools that are already defined are automatically remapped to new tools as needed.

Note: This command requires that you ensure the critical format items (mode, m.n and trailing zero suppression) of the file being imported are correctly specified.

Drill Tool List

This command allows you to import a Drill Tool List file into GerbTool. Specify the file to convert and choose the tool list format using the pull down list. Tool list conversion is accomplished using the same ACR rule file technology as used by the Aperture List import command. To add additional tool list converters, copy the associated conversion rule file into the “ToolConv” folder within the main GerbTool installation folder. See Chapter 8 for more information about ACR files.

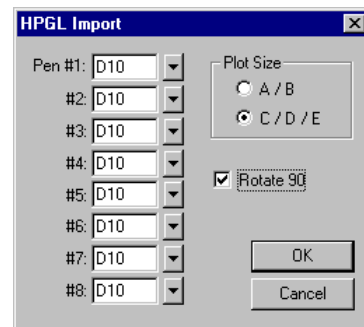
BARCO DPF

This command allows you to import one or more BARCO DPF files into the current design. This command begins importing the specified files into the active layer if it is empty. If it is not empty, the first unused layer following the active layer will be used. GerbTool will use as many layers as necessary to import all the files you specified.

Note: To output the layer in BARCO DPF format use the File/Export/BARCO DPF command.

HPGL

This command allows you to import an HPGL plot file into the currently active layer. After selecting a file to import you will be presented with the following HPGL import dialog box.



HPGL Import Form.

Using the HPGL input form you may specify the expected plot size, whether to rotate the plot data and what D-Codes to use for each HPGL pen.

IPC-D-356

This command allows you to import and associate an IPC-D-356 format netlist into your design. Since an IPC-D-356 formatted netlist contains test point information pertaining solely to pads and not traces, GerbTool must generate an internal netlist prior to importing an IPC-D-356 netlist to ensure that your database contains a full and complete netlist after importing. While this may sound somewhat redundant, the added benefit of an “automatic netlist comparison” is well worth it. The netlist comparison feature produces a report file detailing any differences between the internal and imported netlist. Optionally, the database UserData fields may be updated with the component/net data from the IPC-D-356 file. This allows you to use GerbTool commands, including the Query/Item, Query/Net and Query/UserData commands, to examine and manipulate the true reference designators, pin numbers, etc.

When importing IPC-D-356 files, GerbTool associates the net information for each layer in the input file to the conductive layers of the current design. Occasionally, you may encounter an IPC-D-356 file that contains probe information for some of the inner layers. If you have a file of this type, you should make sure the appropriate layers are

Chapter 7

loaded in the current design. The layers will be associated in the same order they are encountered in the IPC-D-356 file.

GerbTool will optionally create a pad for each test point in the input file. These pads are based on the size and location of the testpoints and will be placed on the specified layer. It is recommended that you specify an empty layer if you enable this option.

GerbTool converts the IPC information into Userdata attached to the pads and traces in the Gerber file. For pads, the format is "netname:component(pin)". For traces, only the netname is attached.

Below is a list of possible error message that can come from importing an IPC-D-356 file:

```
No IPC data for location 2.8750, 3.7500 Layer:1
```

There is a pad on this layer that does not have any matching IPC information.

```
No Gerber data for location 1.5980, 4.3800 ID 45:() idx 43
```

There was an IPC-D-356 record for this location, but no Gerber data.

```
Gerber Net Re-assignment: GerbTool net 78
                        Locations: 1.7980,0.8300 and 2.7980,4.2800
                        IPC nets 55:() 171:()
```

The IPC file has tried to associate the 2 nets, "55:()" and "171:()", to the GerbTool net number 78

```
IPC Net Re-assignment: GerbTool nets 123 250
                        Locations: 2.0980,1.0300 and 3.7980,4.3800
                        IPC net 78:()
```

The IPC file has tried to give the same net information "78:()" to the GerbTool nets 123 and 250.

DXF

This command allows you to import a DXF file into your design. Each layer contained within the DXF file can be mapped to one or more GerbTool layers. This flexibility allows for duplicating information onto multiple layers when, for instance, a pad master layer is used which needs to be merged onto each layer containing traces. Likewise, more than one DXF layer may map to a single GerbTool layer. Layers may be mapped by color so that items of the same color are merged together onto a single GerbTool layer. This feature can be useful for viewing DXF files containing many colors or items which don't share the same color as the DXF layer in which they appear. Blocks may also be mapped to apertures manually by the user or, when the file was exported from GerbTool, automatically when the Auto Map feature is used. Blocks not mapped to apertures are automatically exploded into their individual draw components. Whenever possible it is recommended that blocks be mapped to equivalent apertures as this will significantly ease editing of the design and decrease the size of the database.

Standard SHX font files and SHX Unifont files are supported both for text and shape entities. If text within the DXF file refers to a font which is not present on your system or the font file is of an unrecognized type, a standard font will be used in its place.

Input File

Specifies the DXF file to import.

Report File

Specifies the report file to generate.

Font Directory

Specifies the directory in which SHX font and shape files are to be found.

Source DXF Layer

Specifies the current DXF layer the user has selected to map to zero or more **Destination Layers** in GerbTool.

Destination Layer

Specifies the layers that the user has selected to receive the contents of the currently selected **Source DXF Layer**. Note that more than one layer can be selected by keeping the Ctrl key pressed during selection.

Chapter 7

Map All to Current

Specifies that all DXF layers are to be merged into the current GerbTool layer.

Map Sequentially

Displays a dialog box allowing you to sequentially map DXF layers to GerbTool layers. You are given the option to exclude DXF layer 0 from the mapping. From this dialog box you may also specify the first GerbTool layer to receive DXF layer information.

Map Layers by Color

When checked specifies that DXF file items are to be mapped onto GerbTool layers based on color. Items of color 1 (red) will appear on GerbTool layer 1, those of color 2 (yellow) appear on GerbTool layer 2, and so on. When this option is used make sure the Max Layers setting in the General Configuration Options is set high enough to allow for the highest color expected; otherwise items of a color higher than this will appear on the last (highest-numbered) GerbTool layer. Many drawings use only colors in the range of 1 through 9; however, valid colors can be of any value in the range of 1 through 255.

Scale Factor

Specifies the scale factor used during merging. The default scale factor is 1.

Metric

Check this if your DXF file is in millimeters. Otherwise, inches are assumed.

Line Width

Specifies the line width, in inches, used for zero-width lines. The default width is 0.01 inches.

Note: Zero-width, closed polylines create filled polygons in GerbTool.

Place at Origin

When checked specifies that the DXF design is to have its lower-left corner appear at the origin.

Clear Merge Layers

When checked specifies that all GerbTool merge layers are to be emptied prior to receiving DXF information.

Create Apertures

When checked specifies that apertures used for drawing lines are to be created when an equivalent aperture does not exist. If not checked the next smaller aperture is used. If a next smaller aperture does not exist, then the smallest is used.

Map Blocks

Displays a dialog box which allows you to map blocks in the DXF file to apertures in GerbTool. If Auto Map is selected then all blocks to be mapped must have its name constructed in the same manner as GerbTool DXF Export constructs block names. If Clear Map is selected, then all block mapping associations are removed.

Mill / Rout

This command allows you to import a NC Mill/Rout file, on disk, into the current Mill layer.

Export

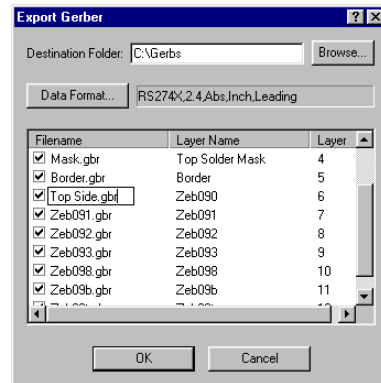
The Export command presents a sub-menu with the following commands:

- | | | |
|-------------------|--------------|---------------|
| ▪ Gerber | ▪ IPC-D-350 | ▪ Bitmap |
| ▪ Aperture List | ▪ IPC-D-356 | ▪ Mill/Rout |
| ▪ Drill | ▪ BARCO DPF | ▪ GerbTool V8 |
| ▪ Drill Tool List | ▪ HPGL | |
| ▪ DXF | ▪ PostScript | |

Gerber

This command allows you to export one or more Gerber files from the current design. You choose the destination folder, select which layers should be exported, adjust their filenames and setup the desired file format.

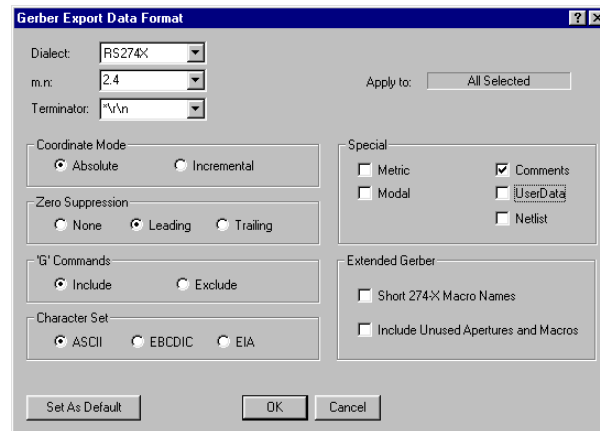
Chapter 7



Export Gerber dialog.

Note: Changing the filenames in the layer list is just like changing the name of a file in the Windows Explorer. Clicking a second time on a filename changes the name to an edit box where you can make your desired changes.

Clicking on the Data Format button will pop up the following dialog allowing you to specify the format of the exported files.



Gerber Export data format dialog.

Dialect

Indicates the specific dialect of the Gerber language such as RS274D, RS274X and FIRE9XXX.

m.n

Coordinate Format such as 2.3. This specifies 2 decimal digits before an implied decimal point and 3 following. (i.e. 12250 represents 12.250 if the coordinate format is 2.3).

Terminator

Indicate the desired block terminator (EOB). Usually this an astrisk. Use ‘\r’ to indicate a carriage return (ASCII 13) and ‘\n’ to indicate a line feed (ASCII 10). Normally, DOS is “*\r\n” and Unix is “*\n”

Coordinate Mode

Choose Absolute or Incremental (See Glossary for descriptions of these terms).

Zero Suppression

Indicate whether leading zeros are suppressed, trailing zeros are suppressed, or no zero suppression.

‘G’ Commands

Indicate whether ‘G’ commands (i.e. G01) should be included when exporting Gerber files.

Character Set

Indicate the expected character set.

Special

You may enable Modal mode which will reduce the size of your files by removing all redundant draft codes and coordinates, enable the saving of G04 comments, enable Metric mode indicating that your files are in metric format, enable the exporting of UserData information, and enable the exporting of netlist information embedded within the Gerber file(s). If you have previously exported a Gerber file with netlist information, you may remove it by disabling the Netlist check-button and re-exporting.

Short 274-X Macro Names (Extended Gerber)

Checking this button forces GerbTool to export all 274-X macro names using a 7 character limit. This setting is for compatibility with plotters that have limited support for the 274-X extended Gerber format.

Include Unused Apertures and Macros

If this setting is enabled, all aperture definitions and macros will be exported whether they are used or not.

Chapter 7

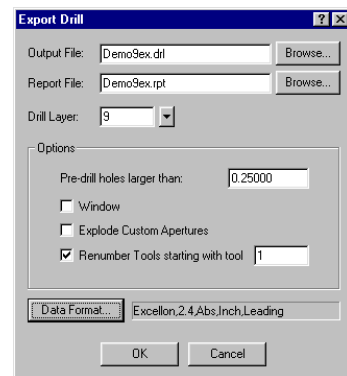
Aperture List

This command allows you to export an Aperture List file from GerbTool. Only GerbTool format is supported.



Drill

The Export Drill command creates a NC Drill file in the format specified. The specified drill layer is presumed to be optimized prior to executing this command (See Tools/Drill/Optimize later in this chapter). Duplicate hits within a single tool are removed.



Drill Export Dialog.

Note: Panelization of the image should be performed prior to executing this command. If you perform a virtual panelization the output of this command will contain step & repeat codes.

Pre-drill holes larger than

Any holes larger than this value will be pre-drilled.

Window

Check if you would like to export only a portion of the drill layer.

Explode Custom Apertures

Indicate how to handle custom apertures. If checked, all data within custom apertures will be output as individual hits. Otherwise, step & repeat will be used to output the custom apertures.

Renumber Tools starting with tool

If checked, the exported tools will be sequentially renumbered starting at the specified tool number.

Drill Tool List

This command allows you to export a Drill Tool List file from GerbTool. Only GerbTool format is supported.

BARCO DPF

Designs exported to BARCO DPF format will be output into a separate file for each layer. You select which layers to export and what the output filenames will be. If you enable the Auto Rename button GerbTool will output all selected layers renaming each layer automatically using the filename extension specified in the New Extension field.

IPC-D-350

Designs exported to IPC-D-350 format will output into one disk file containing all layer data specified within the currently loaded design. The specified output file will contain all data necessary to reproduce your design on any IPC-D-350 compatible device.

IPC-D-356

Designs exported to IPC-D-356 format will output into one disk file containing all layer data specified within the currently loaded design. The specified output file will contain all netlist data associated with the current design.

Output File

Enter the name of the desired output file. Use the Browse button to choose a specific file and/or folder.

Through-hole Drill Layer

Enter the through-hole drill layer number that was used when the netlist was generated. If no drill file is available, the aperture list “type” field will be used to determine whether a pad is SMT or not.

Include unconnected pads using N/C net name

If checked, all unconnected(isolated) pads will be output using the IPC defined net name “N/C”. Otherwise, all unconnected pads will be ignored.

Include soldermask information

If checked, each output record will include data indicating soldermask usage.

Chapter 7

DXF

This command allows you to export your GerbTool design to a DXF file. Each GerbTool layer creates a corresponding DXF layer. In addition a DXF layer 0 is created containing items which appear within the Blocks section. The Blocks section is generated containing blocks with information necessary for displaying each of the apertures used in the design. The user is not required to acquire an equivalent set of blocks for reproducing the apertures which can appear within GerbTool. Note that as DXF does not support the concept of polarity, negative polarity areas within custom apertures will not appear correctly when the file is imported into other applications. Block names are created with a convention which allows for easy import back into GerbTool when the DXF Import Auto Map feature is used. Each pad in the design is output into the DXF file as a block insert. Outputting the pads as references in this manner instead of duplicating the draws for each instance can significantly reduce the size of the generated file.

Output File

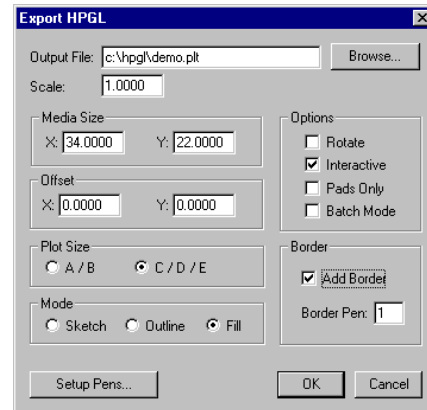
Specifies the DXF file to create during export.

Report File

Specifies the report file to generate.

HPGL

GerbTool provides 3 modes of output when plotting on a HPGL compatible plotter; Sketch, Outline and Fill. Sketch mode is the fastest but does not show width on draws and some flashes such as Donuts. Outline mode shows true width on all objects but they are outlined only. Fill mode shows true width and all objects are completely filled in as they would appear on a photoplot. Fill mode is the slowest and is extremely hard on plotter pens.



HPGL Parameters Form.

You may also specify output file, media size, plot offset, pen width, pen speed, pen number for flashes and draws, pen number for the optional border, scale, whether to rotate 90° and whether to plot only pads (flashes). The offset values are applied independent of the scale specified. Plot offsets allow you to plot multiple images on one sheet.

Add Border

This option adds a border to your plots. To control what text GerbTool adds to this border see 'Options/Configure' later in this chapter for a description of the 'Print Border Text' configuration parameter.

Batch Mode

This option instructs GerbTool to output each visible layer to a separate output file. During batch mode operation, if the Output File field is empty, the output filenames will be derived from the filename associated with each layer and the currently configured HPGL filename extension (see Options/Configure later in this chapter). If, on the other hand, the output file field contains a filename, GerbTool will append a number representing the number of the input layer i.e. demo.001, demo.002, and so on.

Enabling Interactive mode allows you to interactively position each layer on the output page. To position an image on the page, simply click your mouse over an image to select it and then drag the image to the desired location and release the mouse button (or click again).

During interactive plot positioning, a menu of buttons is provided along with several plot specific nested commands.

Chapter 7



HPGL Interactive Control Form.

The Plot button will save the page layout and plot the data. The OK button will save the page layout and quit the interactive session without plotting. And finally, the Reset button allows you to reset the images to their initial positions for this session (if the form has been pinned) or simply quit the interactive session without saving the page layout or plotting the data.

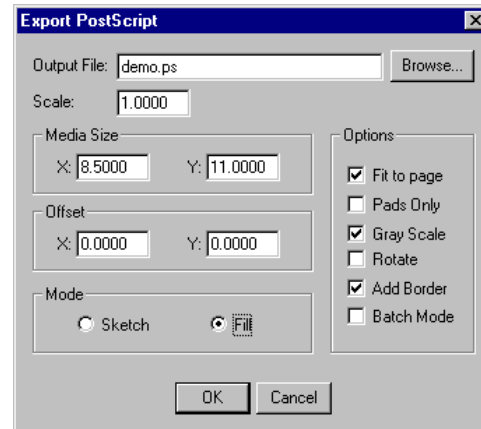
The nested commands available during a interactive plot session are 'C' for absolute coordinate entry, 'T' for page layout initialization, 'L' to cycle the currently selected layer forward, Ctrl+L to cycle the currently selected layer backward, 'S' to snap (align) the currently selected layer on top of another layer and finally, 'R' to redraw the page layout.

Note: There are two files within the GerbTool program directory that affect each HPGL plot. The files 'hpgl.ini' and 'hpgl.dei' are prefixed and appended, respectively, to the actual plot output. If you have any special requirements, you may edit these files as needed.

PostScript

GerbTool provides PostScript output allowing you to plot your data on any device that supports PostScript. This includes typesetters capable of producing production quality artwork. Two modes of output are provided when outputting PostScript; Outline and Fill. Outline mode shows true width on all objects but they are outlined only. This allows you to check for overlapping features. Fill mode shows true width and all objects are completely filled in as they would appear on a photoplot. Fill mode may produce a larger output file.

Enabling Gray Scale mode allows you to output accurate black and white composites as well as halftoned images. When Gray Scale mode is disabled, all colors other than the background color are printed black. When enabled, all colors (other than black/white) are converted to a different gray scale.



PostScript Parameters Form.

You may also specify output file, media size, plot offset, scale including fit to page, whether to rotate 90° and whether to plot only pads (flashes). The offset values are applied independent of the scale specified. Plot offsets allow you to position the image anywhere on the media.

Add Border

This option adds a border to your plots. To control what text GerbTool adds to this border see 'Options/Configure' later in this chapter for a description of the 'Print Border Text' configuration parameter.

Batch Mode

This option instructs GerbTool to output each visible layer to a separate output file. During batch mode operation, if the Output File field is empty, the output filenames will be derived from the filename associated with each layer and the currently configured filename extension (see Options/Configure later in this chapter). If, on the other hand, the output file field contains a filename, GerbTool will append a number representing the number of the input layer i.e. demo.001, demo.002, and so on.

Bitmap

This command allows you to export the current view to a Windows BMP bitmap file. This is the standard bitmap file format that virtually all graphics and page layout applications use.

Chapter 7

Resolution

Specify the resolution in dots per inch (DPI) you wish to use for the output bitmap file.

Black & White

Check if you wish to have the bitmap file created with only black and white colors.

Background Color

Specify the color of the background you wish to have used for the output bitmap file. This option is not available for black and white bitmaps.

Mill/Rout

This command allows you to export a milling layer to a file.

Pre-drill holes larger than

Any holes larger than this value will be pre-drilled.

Window

Check this box if you would like to export only a portion of the selected mill layer.

Explode Custom Apertures

Indicate how to handle custom apertures. If checked, all data within custom apertures will be output as individual hits. Otherwise, step & repeat will be used to output the custom apertures.

Convert Arcs to Line Segments

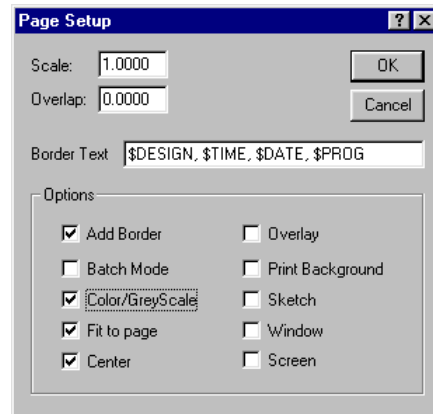
Check if you wish to have arcs converted into line segments prior to exporting.

GerbTool V8

This command allows you to export your design to the GerbTool version 8 format. To use this command successfully, the design data should be exported to a single folder immediately prior to executing this command. The Export/Gerber command should be used and if exporting basic 274D Gerber files, the Export/Aperture List command should be used also.

Page Setup

This command allows you to configure how GerbTool will format each page printed by the Print command.



Page Setup dialog box.

With this dialog box you specify the scale, including fit to page, window mode, color or gray scale, whether to print the background color, sketch or overlay mode and whether batch mode is desired. If batch mode is enabled, each visible layer will automatically be sent to the printer as a separate print job.

The Overlap field allows you to indicate how much to overlap the pages of a multi-page plot to allow proper alignment when taping the pages together.



Print

Select this command when you wish to print the viewed layers to the current Windows printer. This command allows you to print your design on any printer/plotter supported by Windows.

Print Preview

This command allows you to view how each page of your design would print on the current Windows printer before actual printing begins using the Print command discussed above.

Printer Setup

This command allows you to select and configure the current Windows default printer prior to using the Print command discussed above.

Send

If your system has a MAPI enabled email program installed, this command will create a mail message with the current GerbTool design as a email attachment. All you need to do is address the message and provide any additional text.

Exit

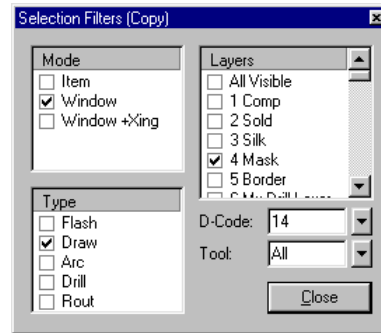
Select this command when you wish to exit GerbTool. The current design file, may optionally be saved and you will be prompted to confirm quitting if any layers have been modified.

Edit Menu

The Edit menu selection presents a sub menu of editing commands. The menu selections include:

- | | | |
|----------|----------|----------------------|
| ▪ Undo | ▪ Clip | ▪ Change Drill Tools |
| ▪ Select | ▪ Join | ▪ Align Layers |
| ▪ Item | ▪ Rotate | ▪ Snap Pads |
| ▪ Copy | ▪ Mirror | ▪ Origin |
| ▪ Move | ▪ Scale | ▪ Purge |
| ▪ Delete | ▪ D-Code | |

All editing commands that require you to modify one or more database items, will allow you to control the 'Selection Filter' for determining what database items to select or modify. GerbTool commands are very flexible in the selection of data to modify. For example, depending on the command, you may choose from single item, window, group or complete layer selections as well as restricting your selections to particular layers and/or D-codes.



Selection Filter.

All editing commands may be terminated by clicking the right button, touching the escape key or selecting another menu item.

Note: See Chapter 4, 'Nested Commands' for details on using GerbTool nested commands. Nested commands are selected with one key stroke and operate immediately, even during another command.



Undo

This command allows you to fully 'undo' changes you've made to the currently loaded database. Undo information is saved in a 'last in - first out' fashion. This means that you 'undo' changes in the reverse order in which the changes were made. This allows you to undo the most recent changes first. You may also use the nested command 'u' to undo the most recent command even during another editing command.

Note: Undo must be enabled with the Options/Configure command prior to making any edits if you plan to use this command.

Note: Undo increases the amount of memory GerbTool requires. If you do not require the undo capability, you may disable undo with the Options/Configure command. Disabling undo will release any memory currently associated with undo information and prevent further undo memory use.

Select

Most editing commands (such as Copy, Move etc.) allow you to work with single items, windows of items or groups of items. The commands available in the Select menu allow you to manage the grouping of items for use by these editing commands. When a command allows group selection mode, it will use the currently selected group created and maintained by the different Select commands. Select groups are also persistent from one command to another. For example, if you rotate the current select group, the rotated data will remain selected ready for another command.

The Select menu contains the following commands for controlling select groups:



New Group

This command allows you to start a new group of selected items. You will be prompted for confirmation to clear the current select group if any. This does NOT destroy any data. It simply un-selects the current select group. If you respond affirmatively, the Group Selection Filter dialog will be presented and you will be automatically placed in the Select/Add To command.



Add To

Use this command to select more items and place them in the current select group. The Group Selection Criteria form will be presented where you specify what types of items you would like to select.



Remove From

Use this command to selectively remove items from the current select group. The Group Selection Criteria form will be presented where you specify what types of items you would like to un-select.

Invert

Use this command to invert the current select group. That is, all currently selected items are un-selected and all un-selected items become selected. One use of this command is to allow you to quickly select all but a few items by first selecting the items you don't want and then inverting the select group.



Off

Use this command to clear the current select group and un-highlight any highlighted items. This does NOT destroy any data. It simply un-selects the current select group.



Item

This command displays a floating dialog box that displays and allows you to edit pertinent information associated with each database item. In addition to allowing you to edit each database item there are extensive controls for navigating from one item to another including the ability locate an item based on its sequential position in the database, D-Code or Tool, X-Y coordinate, net and UserData value. You may also step forward and backward one item at a time using the supplied directional buttons. You may also use the 'N' hot key to automatically advance to the next item in the database.

Edit Item dialog.

The UserData field is of special note as this field allows you to attach any textual information you would like to each individual database item. Any text you associate with your database will automatically be saved within your Gerber files the next time you save them. This also allows you to pass on this data to other groups in your organization transparently.

An obvious use is to associate actual reference designators, pin numbers and net names with each pad thereby adding intelligence to your Gerber databases. Besides being able to see UserData using the Query/Item command, Macros also have complete read/write access to each UserData field. This allows some pretty powerful tools to be built upon GerbTool.

Chapter 7

Other than a 256 character size limit, there are no other restrictions on what text can be associated with a database item.



Copy

You may use this command to copy single items, windows or groups of items. Click on the 'Destination Layers' button in the lower left portion of the status bar to specify one or more destination layers.

Note: If you select data from more than one layer, i.e. more than one layer is visible, and copy to one or more destination layers, all copied data will be merged into the destination layers. If you do not choose a destination layer then the data will be copied into the respective source layers.

When copying by window, a picture of the data being copied is dragged on the screen along with the cursor when selecting insertion points. If the amount of data is substantial and the cursor appears sluggish, you may constrain the amount of data dragged by holding down the <ctrl> key. This does not alter the amount of data being copied.



Move

You may use this command to move a single item, a window or groups of items. Click on the 'Destination Layer' button in the lower left portion of the status bar to specify a destination layer.

As with Copy above, if you select data from more than one layer, i.e. more than one layer is visible, and move to a destination layer, all moved data will be merged into the destination layer. If you do not choose a destination layer then the data will be moved into the respective source layers.

Also, as with Copy above, you may constrain the amount of data dragged by holding down the <ctrl> key. This does not alter the amount of data being moved.



Delete

Select this command when you wish to delete items from one or more layers. Either vertices, single items, a window or groups of items may be deleted. Click on the 'Disable Prompts' button in the lower left portion of the status bar to disable confirmation prompting. Click the 'Enable Prompts' button in the lower left portion of the status bar to re-enable confirmation prompting.

Note: If Undo is disabled, you must confirm each deletion.



Clip

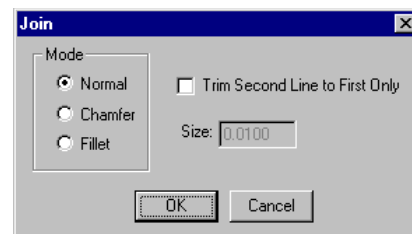
This command provides the ability to specify a window in which all data will be deleted with automatic clipping of draws that pass through the window. If group mode is selected only items within the group will be considered when examining data within the specified window.

Note: The On Boundary selection controls whether flashes that straddle a window boundary are deleted or not.



Join

This command provides the ability to join two line segments together using several different methods.



Join Parameters Form.

Using Normal mode, the two line segments chosen will be extended or trimmed as needed so that they connect. Naturally, this command will not work for parallel or near parallel lines. An option to normal mode, Trim Second Line to First Only, helps when you have a long line in one direction and several lines intersecting the long line. With this option only the second line you select will be modified. The remaining modes, Chamfer and Fillet, use the Size field to determine how far back to trim

Chapter 7

each of the two selected lines before adding the chamfer or fillet accordingly.



Rotate

Use this command when you need to rotate a window or group of items. You may select Window mode or Group mode. You may also supply a pivot point (interactive) or allow automatic calculation of the center of the data (calculated) for the required pivot point.

Note: If the 'Rotate D-Codes' checkbox is checked, new D-Codes may be added to the aperture list.



Mirror

Use this command if you need to mirror (flip) a group of items either horizontally or vertically. GerbTool will allow you to specify the direction to mirror and whether to prompt for the pivot point or automatically calculate it. This command may also be used to flip a secondary side layer that was designed as seen from the primary side.



Scale

This command allows you to apply coordinate offsets and scale to the current design.

The offsets and scale are applied to the selected data. By applying a scale factor it is possible to expand or shrink the size of your database. For example, this command may be used to compensate for shrinkage, or, if your design is at 2X you can set both the X and Y scale factor to 0.5 to convert your files to 1X.

D-Code

The D-Code menu item presents a sub-menu with the following commands:

- Transcode
- Expand
- Scale
- Polarity



Transcode

This command allows you to change the D-Code of an individual item, window, group or complete layer. By changing the D-Code of an item, you can alter its size and shape. Another way to change an items size and shape is to edit the aperture list directly. Click on the 'New D-Code' button in the lower left portion of the status bar to specify the current replacement d-code.

Expand

Use this command to expand custom apertures assigned to one or more dcodes in a design. All data within the selected custom apertures will be expanded and converted into individual data items such as flashes, drills and draws. For additional information on custom apertures see the Tools/Convert/To Custom command later in this chapter also.



Scale

Use this command to shrink or expand the size of one or more D-Codes. One use of this command is to create soldermasks automatically. GerbTool will add new apertures to the corresponding aperture list as needed based on your specified scale factor. If the Fixed Amount check button is enabled, the scale values will be added to each D-Code. Otherwise, each D-Code size will be multiplied by the scale values specified.

Hint: Creating a soldermask is a simple and easy process using this command. First create the soldermask layer by copying the pad master layer onto a new layer. Use Edit/Copy command to copy the pad master to the new layer. When copying, select an empty layer from the 'Destination Layers' list. Then, select this command, enter a scale factor for both X & Y and click on the Fixed Amount field. in the D-Code Scale dialog and click on the OK button. GerbTool will add apertures to the aperture list as necessary and replace the D-Codes with the new scaled D-Codes. The original D-Codes within the aperture list are not modified.



Polarity

Use this command to control the item level polarity of BARCO format files as well as FIRE9XXX raster fill polygons. When using item level polarity, the ordering of the data is crucial. You may find that you need to move data 'in place', thereby placing the 'moved' data at the end of the database.

Note: Gerber (RS-274D) does not support polarity at all. Extended Gerber (274X) files only support polarity at the layer level which is controlled using the Setup/Composites command. FIRE9XXX format only supports raster fill polygons at the item level. Otherwise layer oriented polarity is assumed.



Change Drill Tools

This command allows you to easily change drill tool assignments for existing database items. You may select by item, window, group or layer as well as a specific tool or all tools.



Align Layers

This command allows you to easily and quickly align any mis-aligned layers. Layer registration involves lining up all layers so that when multiple layers are viewed simultaneously they appear correctly aligned. Proper layer alignment is also crucial to the successful creation of a multi-layer netlist.

First determine the layer to which all other layers should be aligned with (a master layer) and select an item to use as a reference point. Select the item you chose as a reference point. Then select an item, on each layer to be aligned, that corresponds to the reference point. As you select each additional item, the entire layer will be automatically aligned.

Note: Remember, you may use the nested zoom in/out and pan keys (see Chapter 4, 'Nested Commands') to make it easier to home in on the reference and corresponding items.

Snap Pads

This command allows you to cleanup low resolution artwork that doesn't line up properly from layer to layer. Unlike the Align Layer command detailed above, this command examines each individual item on a layer and snaps its x/y location to that of the nearest item on the "golden" layer. You supply a tolerance value that will be used to determine whether a particular pad/drill is close enough to a golden layer pad/drill to be snapped to it.

Golden Layer

Select the layer whose pad/drill locations will be used for reference to align items on the Snap Layer(s).

Snap Layer

Enter the layer that contains items to be aligned with items located on the Golden Layer.

Tolerance

Enter the distance that coordinates of items on the Snap Layer can deviate from the items on the Golden Layer to be considered for alignment.



Origin

This command allows you to relocate the origin (0,0 point) of the database. GerbTool will prompt for a point to define the new origin. The film box will be moved to the new origin.

Purge

Use this command to compact the currently loaded database for more efficient use of memory. Since GerbTool doesn't actually remove data from memory during edits, memory may become fragmented and less efficient. Therefore, occasional purging may help GerbTool perform optimally.

Note: Purging destroys any undo information that currently exists. Do not use this command unless you are sure you don't need to 'undo' any previous edits!

View Menu

The View main menu selection displays a menu of commands to control the viewing window location, size and attributes. The menu selections are shown below:

- | | | |
|------------|--------------------|--------------------|
| ▪ Window | ▪ Overlay | ▪ Selection Filter |
| ▪ Zoom In | ▪ Grid | ▪ Save |
| ▪ Zoom Out | ▪ Composites | ▪ Recall |
| ▪ Pan | ▪ Virtual Panel | ▪ Previous |
| ▪ All | ▪ Clear Highlights | ▪ Toolbars |
| ▪ Film Box | ▪ Highlights | ▪ Split |
| ▪ Redraw | ▪ Selections | |
| ▪ Sketch | ▪ Errors | |



Window

This command allows you to select a new viewing window. Two points are required to define a window. The two points define a rectangle that encompass the area that is to become the new viewing window. Use this command when you want precise control over the viewing window.



Zoom In

This menu item halves the size of the current viewing window using a center point that you supply. This command provides a closer look at the displayed data.



Zoom Out

Doubles the size of the current viewing window using a center point you supply. Use this command to quickly increase the size of the viewing window.



Pan

Moves the current viewing window to a new location. The new location is centered about a point you supply. This command does not change the size of the viewing window.



All

This menu item adjusts the size of the viewing window to encompass the extremes of the currently displayed layer(s). No user interaction is required. If data has been deleted from any displayed layers you may need to use the Query/Extents command to calculate the current extremes of the database.



Film Box

Select this command to adjust the size of the viewing window to display the contents of the currently specified film box. This command does not check to see that all data lies within the film box borders. Therefore, depending on the film box size, not all data may be displayed. No user interaction is required.



Redraw

This command simply redraws the current viewing window.



Sketch

This command toggles sketch mode viewing on/off. When sketch mode is enabled, items are shown with an outline only. As an alternative to outline mode, stick mode may be selected using the Options/Configure Display page. This mode can help you spot stacked and buried items.



Overlay

This command toggles overlay viewing mode on/off. When overlay mode is enabled, items become transparent when drawn atop each other. When disabled, items obscure whatever is drawn previously. Overlay mode makes it easier to spot stacked and buried pads. Overlay mode also displays significantly faster.



Grid

This command toggles the grid display on/off. To change the size of the grid and control grid snap mode, see the 'Options/Configure' section later in this chapter.

Note: This command is also available as a nested command by touching the Ctrl+G key (see Chapter 4, 'Nested Commands' for more information about nested commands).



Composites

Enables the correct viewing of composite layers. When this button is enabled the polarity of each layer, specified within the Setup/Composites dialog, will be honored. If a layer is specified 'Clear', all data in that layer will be displayed with the current background color.

Virtual Panel

Virtual panel mode and hence the display of virtual panels may be toggled on/off using this command. You may also use the Ctrl+V nested command. See Chapter 4, 'Nested Commands' for more information about nested commands.

Clear Highlights

This command clears any and all highlights that currently exist. Use this command after a command such as Query/Highlight or Tools/Drill/Show Drill Path.

Highlights

This command toggles the display of normal highlights on/off. Normal highlights are all highlights not indicating a select group or DRC error.

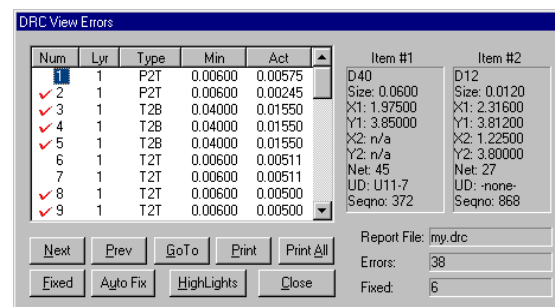
Selections

This command toggles the display of select group highlighting. This does not change the actual select group. Only the highlight display is changed by this command.



Errors

This command is used to view rule violation errors after performing a DRC. If DRC errors exist, the DRC View Errors dialog box shown below will be displayed.



DRC View Errors.

See the Tools/DRC command later in this chapter for information on viewing DRC errors.



Selection Filter

This button toggles the display of the current 'Selection Filter' if any. Many editing commands present the selection filter dialog which you may toggle on/off using this menu item, the appropriate 'Settings' toolbar button or the nested command 'F'.

Save

This command is used to save the current viewing window for later recall. There are eight positions available, 1-8, for saving. The current viewing window will be saved in the position that you click on. Use the View/Recall command to recall any of the saved viewing windows.

Recall

This command is used to recall a previously saved viewing window (see 'Save' above). If any of the eight possible positions does not have a viewing window associated with it, the corresponding position in the sub-menu will be disabled.

Previous

This command is used to recall the last viewing window. This allows you to quickly and easily toggle between to viewing locations.

Toolbars

This menu selection presents a dialog listing all of the available tool bars. You may toggle each toolbar on/off as desired.

Split

This command is used to split the drawing area into multiple panes. By dragging the pane dividers to the desired location you can have up to four separate viewing panes. Each pane may have a different zoom level and/or location allowing you to view and edit multiple views of your design at one time.

Add

The Add main menu selection presents a menu of commands that allow you to enter various types of new database items. This menu is shown below:

- | | |
|-------------|------------|
| ▪ Flash | ▪ Circle |
| ▪ Draw | ▪ Arc Ctr |
| ▪ Rectangle | ▪ Arc 3 Pt |
| ▪ Vertex | ▪ Array |
| ▪ Drill | ▪ Polygon |
| ▪ Slot | ▪ Text |

Note: All circles and arcs are created using circular interpolation or with multiple line segments depending on the style indicated by the Options/Arcs 360 command. Use circular interpolation with care as not all photo-plotters support circular interpolation. Segmented circles and arcs use the chord angle specified using the Options/Configure command.



Flash

This command allows you to add a flash to the active layer. GerbTool prompts for a point at which to add the flash. As you move the cursor about the screen an outline shape of the current D-Code is displayed. Click left to add a flash at that location.



Draw

This command allows you to draw line segments in the active layer. GerbTool will prompt for a starting point and subsequent points to form continuous traces. Click right or touch the [Esc] key to start a new trace. To add arcs, push the 'A' key on your keyboard. This will enter arc mode and allow you to enter a 3 point arc. After entering the end point of an arc you may push the '9' key for an automatic 90 degree arc.



Rectangle

This command allows you to draw line segments in the shape of a rectangle to the active layer. GerbTool will prompt for a starting corner point and a opposite corner point.



Vertex

This command allows you to add (and move by dragging the mouse) a vertex anywhere on an existing line segment.



Drill

This command allows you to add a drill hit to the active 'Drill' layer. GerbTool prompts for a point at which to add the drill. You may click the mouse or use absolute/relative coordinates from the XY toolbar. The current tool, as displayed in the layer toolbar, will be used.



Slot

This command allows you to draw drill slots in the active 'Drill' layer. GerbTool will prompt for a starting point and subsequent points to form continuous slots. Click right or touch the [Esc] key to start a new slot. The current tool, as displayed in the layer toolbar, will be used.



Circle

This command allows you to draw a circle by entering a center point and a point on the radius. The circle is drawn on the active layer, using the current D-Code, in a counter-clockwise direction. Please see the note at the beginning of this section concerning how circles are created.



Arc Ctr

With this command you define an arc by entering a center point, a point defining the radius and starting angle, followed by a point defining the

Chapter 7

ending angle. The arc is drawn on the active layer, using the current D-Code, in a counter-clockwise direction. Please see the note at the beginning of this section concerning how arcs are created.



Arc 3 Pt

With this command you define an arc by entering its end points and then a point on its circumference. The arc is drawn on the active layer, using the current D-Code, in a counter-clockwise direction. Please see the note at the beginning of this section concerning how arcs are created.

Note: To create 90° arcs easily and quickly, touch the '9' key. This will automatically construct a 90° degree arc.

Array

This command allows you to add an array of items to your database. GerbTool will display the Add Array dialog where you may edit the pattern spacing.

You may then define a rectangular area by entering two points. As you drag the second corner of the rectangle, the current number of rows and columns along with a total will be displayed in the status bar. GerbTool will fill the specified area with a pattern of dcodes or tools, depending on the layer type of the active layer.



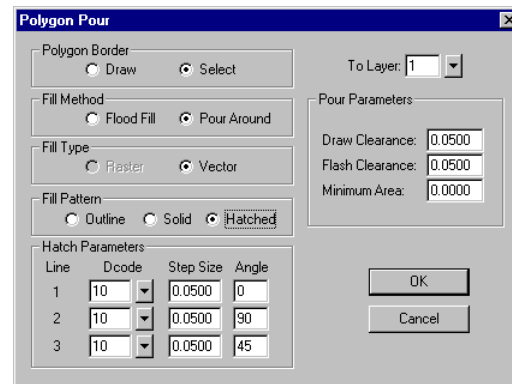
Polygon

This command allows you to select or enter a closed polygon and GerbTool will fill/pour the interior of the polygon using either a raster fill or vector fill method. This command is commonly used to create ground plane areas.

Note: Raster filling is not supported in RS274D format Gerber files.

When entering a polygon you may automatically close the polygon by touching the [End] key at anytime. You may also close the polygon manually by entering a point at the point which began the polygon. Regardless of the method of closure, GerbTool will then outline the polygon with the current D-code, as displayed in the status bar, and begin filling the interior of the polygon. In Flood Fill mode, GerbTool will fill the interior of the polygon with increasing aperture sizes. As it fills toward the center of the polygon the aperture sizes will become larger and larger. In Pour Around mode, GerbTool will fill the interior

of the polygon, as above, while maintaining clearance, as specified by the Draw Clearance and Flash Clearance fields, around all circuitry.



Polygon edit dialog.

As many smaller polygons may be generated to effectively "pour" around the circuitry, the Min Area parameter allows you to specify the minimum size area that will be allowed. Any filled areas that would be smaller than the specified Min Area will be automatically eliminated.

The Pour Around option also supports three additional modes: Outline, Solid and Hatch mode.

If Outline mode is selected no filling of the resultant polygons takes place. This type of output may be used to drive PCB prototyping equipment.

If Solid mode is selected the resultant polygons are filled completely using the same methods described for the Fill command above.

If Hatched mode is selected the resultant polygons will be filled with a cross hatched pattern as specified in the Hatch Parameters section of the edit form. Up to three lines may be used with different sizes and angles for each line.



Text

The Text command provides the ability to insert text into the database as a sequence of line segments. Therefore, you can control the line thickness of the inserted text by changing the current D-Code. Text may be rotated, mirrored or slanted if desired. The height and width of the

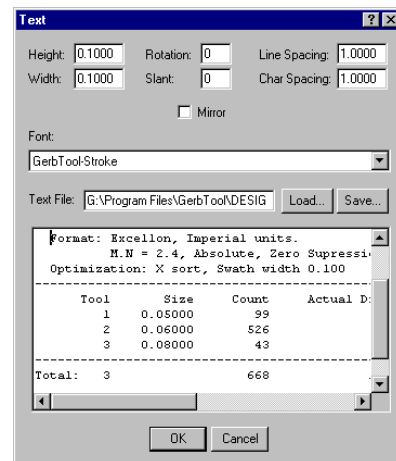
Chapter 7

text is also user specified as is the inter character and line spacing. You may select any font from the Font pulldown list. This list contains all the TrueType fonts on your system and the special “GerbTool-Stroke” font. This special font is installed with GerbTool and is a simple font that does not use polygonal data or negative polarity.

The following is a list of important points to remember when using TrueType fonts:

- TrueType fonts require the use of polygonal data and a combination of positive and negative polarity. By nature, only RS274X and FIRE9000 extended Gerber file formats support negative data and polygonal data.
- GerbTool modifies the layer setup by adding additional composite layers. Any previously generated report files that specify layer numbers will be subsequently out of sync.
- As TrueType fonts require the use of composites layers, composite viewing must be enabled to view the text properly. While running this command enables this mode automatically, you may use the View/Composites command to toggle this mode on and off.

A text editing window is provided where you may enter as many lines of text as needed. You have full editing and scrolling capabilities. You may even load and save text files.



Add Text Dialog.

Setup Menu

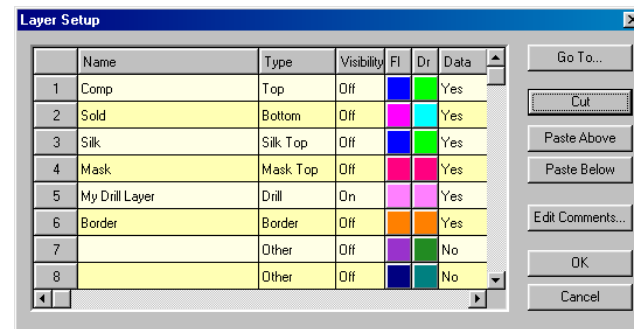
The Setup main menu selection displays a menu of commands for managing the different tables that GerbTool utilizes.

- Layers
- Apertures
- Drill Tools
- Mill Tools
- Composites
- Layer Sets



Layers

The Layers command displays and allows you to edit the layers dialog.



Layer Setup Dialog.

Following is a description of each field within the Layers dialog:

Name

Descriptive name of layer. This is not necessarily a filename.

Type

Allows you to specify a layer type of Top, Inner, Bottom, Plane, Silk Top, Silk Bottom, Mask Top, Mask Bottom, Border, Drill, Mill, Composite or Other.

Note: It is important to properly specify what layer type each layer is as several GerbTool commands check this field for the proper type before processing each layer. For example, the Tools/Pad Removal command will only operate on layers with a type of Inner.

Visibility

Controls the visibility of the specified layer. Options are On, Off or Ref.

Chapter 7

Flash/Draw

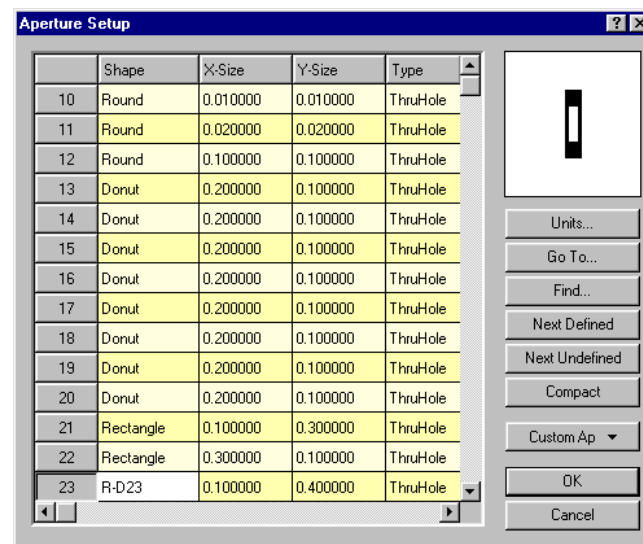
Color buttons control the color of flashes and draws respectively.

The Cut, Paste Above and Paste Below buttons allow you to re-order the layer structure. In addition, if you cut a layer from a loaded design without pasting the layer, you will be prompted if you would like to unload that layer from memory upon clicking the OK button.



Apertures

This command allows you to edit the aperture list. Columns can be sorted by clicking on the appropriate column heading. An asterisk (*) will appear next to each D-code that is currently used in the design.



Aperture Edit Dialog.

Shape

This pull down allows you to select the desired shape. All custom apertures currently loaded will be included in this list.

X/Y Size

When editing the X-Size field, if the Y-Size field contains 0.0 and the Shape field specifies a symmetrical shape, it will be set to the value of the X field automatically. For non-symmetrical shapes such as Rectangle and Oblong, the X size is the major size and the Y size is the

minor size. For Thermals and Donuts, the X size is the outside diameter and the Y size is the inside diameter. For Targets, the X size is the outer ring diameter and the Y size is the inner ring diameter.

Type

This field specifies whether this D-Code represents a surface mount, through-hole, or thermal pad.

Note: If a drill layer is specified during multilayer netlist generation (see Chapter 7, ‘Tools/Netlist’), this field does not need to be set except for thermals. If your design has internal planes that use custom apertures for thermals, this field must be set to ‘Thermal’.

Units

This button pops up the Options/Configure/Units and Precision dialog. Any changes you make to this dialog will be reflected in the X and Y fields when you ‘OK’ the dialog.

Go To

This button allows you to go directly to a particular dcode.

Find

This button allows you to search for an aperture that contains the text string you specify. Any text appearing in the scrollable aperture list may be searched for. For example, you could type ‘rect’ to find the next occurrence of a Rectangular aperture. You could also enter ‘.05’ to find the next occurrence of a 50 mil aperture.

Compact

This command allows you to remove unused and redundant apertures within the aperture list. Each layer then has it’s D-Codes remapped accordingly.

Custom Ap

This button pulls down a menu with the following commands for manipulating custom apertures:

Edit

This command allows you to select and edit a currently loaded custom aperture. After selecting the desired custom aperture, the Custom Aperture Editor will pop up with the selected custom aperture loaded ready for modification. After making your changes, click the Custom Aperture Editor File/Save command to update the GerbTool copy.

Chapter 7

New

This command allows to create a completely new custom aperture. After supplying a name for your new custom aperture the Custom Aperture Editor will pop up. You may select the newly created custom aperture from the aperture Shape pull down list.

Delete

Use this command to delete a currently loaded custom aperture.

Rename

Use this command to rename an existing custom aperture.

Load From Lib

This command allows you to load one or more custom apertures from an external custom aperture library file or GerbTool design file. You select which apertures to load. After loading, you may select the newly loaded custom apertures from the aperture Shape pull down list.

Save To Lib

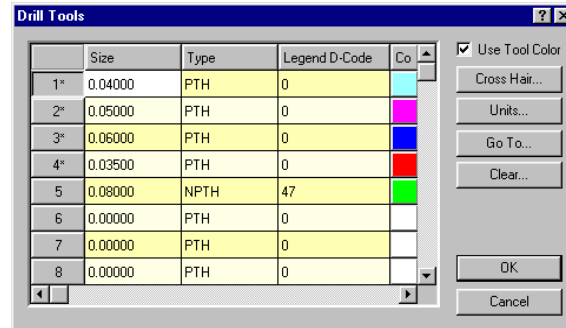
This command allows you to save one or more currently loaded custom apertures to an external custom aperture library file.

WARNING: This command does NOT merge the customs into existing library files. All original data within the chosen library file is cleared before saving.



Drill Tools

This command allows you to edit the drill tool list. An asterisk (*) will appear next to each tool that is currently used in the design. Columns can be sorted by clicking on the appropriate column heading. All drill layers will updated with the new tool number ordering when you click the OK button.



Drill Tool Edit Dialog.

Use this command to edit the list of drill tools to be used when handling drill layers as well as importing/exporting drill files.

Size

Enter the size of the drill tool here.

Type

Indicates whether a tool is to be used for plated or non plated holes.

Legend D-Code

This D-Code will be used when displaying the drill layer. If a D-Code is not assigned, a circle will be displayed using the size of the tool.

Color

You may also assign a color to each individual tool. These colors, if enabled with the 'Use Tool Color' button, will override the layer colors already assigned.

Cross-Hair

This button pops up a dialog where you can set the properties of the drill cross-hair displayed during redraws.

Units

This button pops up the Options/Configure/Units and Precision dialog. Any changes you make to this dialog will be reflected in the X and Y fields when you 'OK' the dialog.

Go To

This button allows you to go directly to a particular tool.

Chapter 7

Clear

Click to clear the tool list. You will be warned if drill tools are referenced in the design.

Mill Tools

This command allows you to edit the mill/rout tool list. Use this command to edit the list of mill/rout tools to be used when handling mill layers as well as importing/exporting mill files.

Size

Enter the size of the drill tool here.

Feedrate

Indicate the desired feedrate of the selected tool.

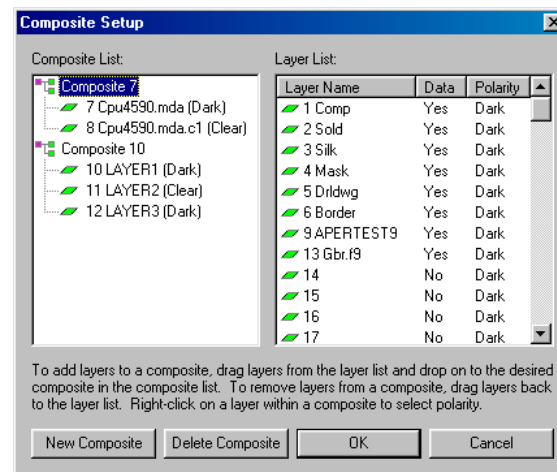
Legend D-Code

This D-Code will be used when displaying the mill layer. If a D-Code is not assigned, a circle will be displayed using the size of the tool.



Composites

This command allows you to create and modify composited layer sets.



Composite Setup Dialog.

Use this dialog to create new composites, delete existing composites and add/remove layers to/from existing composites.

To add a layer to a composite, drag the desired layer from the layer list to the composite list. To remove a layer from a composite, drag the layer back to the layer list.

To change layer polarity, 'right click' on a layer within a composite to select a different polarity.

Note: Not all exported file types support layers that use polarity.

New Composite

Click on this button to create a new, initially empty, composite.

Delete Composite

Click on this button to delete an existing composite.

Note: This command does NOT delete any data or remove any data layers. Only the association to other layers is removed.

Layer Sets

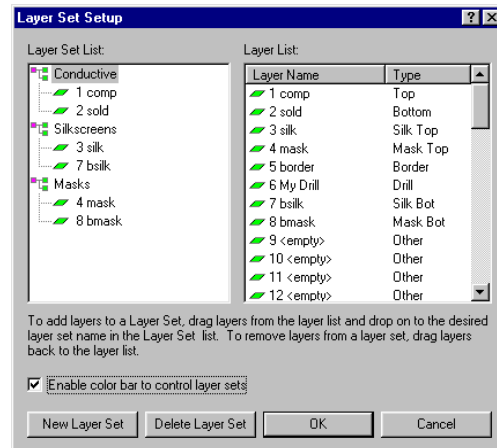
The Layer Sets menu item presents a sub-menu with the following commands:

- View/Edit
- Blind/Buried

View/Edit

This command allows you to create layer sets. Layer sets allow you to set the visibility state of multiple layers, belonging to a given set, simultaneously.

Chapter 7



View/Edit Layer Set Dialog.

Use this dialog to create new layer sets, delete existing layer sets and add/remove layers to/from existing layer sets.

To add a layer to a layer set, drag the desired layer from the layer list to the layer set list. To remove a layer from a layer set, drag the layer back to the layer list.

New Layer Set

Click on this button to create a new, initially empty, layer set.

Delete Layer Set

Click on this button to delete an existing layer set.

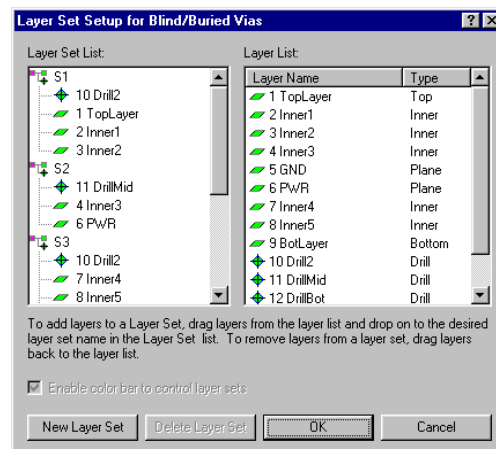
Note: This command does NOT delete any data or remove any data layers. Only the association to other layers is removed.

Enable color bar to control layer sets

Check this option to allow the toggling of the visibility state of selected layer sets from the color dialog bar.

Blind/Buried

This command allows you to create layer sets to define the pairing of drill layers to circuit layers for blind and/or buried vias. If your design includes blind/buried vias, the relationship of drill layers to circuit layers must be defined before generating a netlist using the Netlist Generate command.



Blind/Buried Layer Set Dialog.

Use this dialog to create new layer sets, delete existing layer sets and add/remove layers to/from existing layer sets.

To add a layer to a layer set, drag the desired layer from the layer list to the layer set list. To remove a layer from a layer set, drag the layer back to the layer list.

Important: Each layer set must contain a single drill layer. Drill layers are indicated by the type name of 'Drill' and the drill hit icon.

New Layer Set

Click on this button to create a new, initially empty, layer set.

Delete Layer Set

Click on this button to delete an existing layer set.

Note: This command does NOT delete any data or remove any data layers. Only the association to other layers is removed.

Documentation

The Setup main menu selection displays a menu of commands for managing the different tables that GerbTool utilizes.

- Reports
- Redline
- Drawing

Reports

The Reports menu item presents a sub-menu with the following commands:

- Apertures
- Drill Tools
- Mill Tools



Apertures

Select this commands to generate an Aperture Report. An aperture report details which D-Codes, along with their definitions, are being used on a per layer basis. Included in the report are use counts for both flashes and draws.

Aperture Use Report					
Layer 9: APERTEST9					
Units: Inches					
Dcode	Shape	Width x	Height	Flash	Draw
D14	Round	0.0200 x	0.0200	0	376
D28	Round	0.1000 x	0.1000	16	0
D121	Round	0.0100 x	0.0100	0	1900
D122	Donut	0.2000 x	0.1000	8	0
D123	Rect	0.1000 x	0.3000	8	0
D124	Rect	0.3000 x	0.1000	8	0
D125	R-D125	0.1000 x	0.4000	4	0
D126	R-D126	0.4000 x	0.1000	4	0
D127	Oblong	0.1000 x	0.3000	4	0
D128	Oblong	0.3000 x	0.1000	4	0
D129	Oblong	0.2500 x	0.1000	4	0
D130	Oblong	0.1000 x	0.2500	4	0

Buttons: Edit Selected... Layer: 9 Generate Save... Print Close

Aperture Use Report.

Note: If an aperture has an unknown shape, or is zero in size, it will be highlighted for easy recognition.

Edit Selected

Clicking this button will popup the Aperture list editor with the current aperture set to the aperture highlighted in the report. You may also double click directly on a report item to edit that aperture.

Generate

Each time this button is pressed a report will be displayed for the layer specified in the Layer field. Entering 'all', or a '0', in the Layer field will instruct GerbTool to generate a Combined Aperture Report for all loaded layers. You may use the scroll bar to view all of the report if it does not fit entirely within the window.

Print

You may print the report to the default Windows printer using this button.

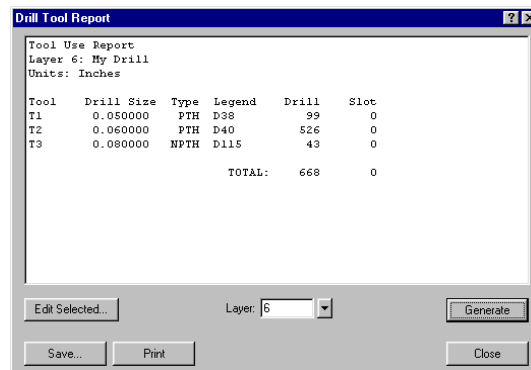
Save

You may save the report to a file for further manipulation or later printing by clicking on this button.



Drill Tools

Select this command to generate a Drill Tool Report. A drill tool report details which drill tools, along with their definitions, are being used on a per layer basis. Included in the report are use counts for both drills and slots.



Tool Use Report.

Note: If a drill tool is zero in size, it will be highlighted for easy recognition.

Chapter 7

Edit Selected

Clicking this button will popup the Drill Tool editor with the current tool set to the tool highlighted in the report. You may also double click directly on a report item to edit that tool.

Generate

Each time this button is pressed a report will be displayed for the layer specified in the Layer field. Entering 'all', or a '0', in the Layer field will instruct GerbTool to generate a Combined Tool Use Report for all loaded layers. You may use the scroll bar to view all of the report if it does not fit entirely within the window.

Print

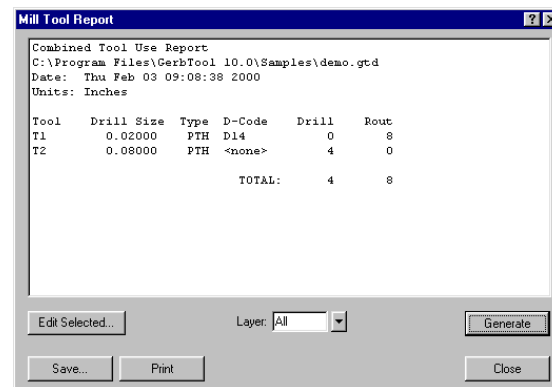
You may print the report to the default Windows printer using this button.

Save

You may save the report to a file for further manipulation or later printing by clicking on this button.

Mill Tools

Select this command to generate a Mill Tool Report. A mill tool report details which tools, along with their definitions, are being used on a per layer basis. Included in the report are use counts for both drills and routs.



Tool Use Report.

Note: If a mill tool is zero in size, it will be highlighted for easy recognition.

Edit Selected

Clicking this button will popup the Mill Tool editor with the current tool set to the tool highlighted in the report. You may also double click directly on a report item to edit that tool.

Generate

Each time this button is pressed a report will be displayed for the layer specified in the Layer field. Entering 'all', or a '0', in the Layer field will instruct GerbTool to generate a Combined Tool Use Report for all loaded layers. You may use the scroll bar to view all of the report if it does not fit entirely within the window.

Print

You may print the report to the default Windows printer using this button.

Save

You may save the report to a file for further manipulation or later printing by clicking on this button.

Redline

The Redline menu item presents a sub-menu with the following commands:

- | | |
|--------------------|------------------|
| ▪ Add Text | ▪ Sketch |
| ▪ Add Balloon Text | ▪ Delete |
| ▪ Add Arrow | ▪ Properties |
| ▪ Add Line | ▪ View Redlining |

The Redline menu provides commands which allow you to edit redline information contained in your design. Use the redlining features to add comments and other information which needs to be stored separately from normal layer information. Every layer may have it's own associated redline information attached to it.

Redline objects are displayed using the current redline color and D-Code. The Options/Configure/Display dialog specifies the default settings when these have not been specified by the user.

When a design is saved the redlining information is stored in a separate file located in the same directory as the design file. The redline file has the same name as the design file, only with 'red' as the filename extension.

Chapter 7



Add Text

This command allows you to insert text into the currently selected layer's redline information. The text size may be specified along with rotation, slant, mirroring and spacing. The text may also be saved and loaded from a file.



Add Balloon Text

This command allows you to insert a balloon with text into the currently selected layer's redline information. Various text attributes may be specified in the same manner as with the Add Text command. After specifying text you wish to have appear in the balloon you specify the point you wish the balloon arrow to point and then the location of the upper-left corner of the balloon box. Repeatedly specifying points will insert multiple balloons containing the same text.



Add Arrow

This command allows you to insert an arrow of specified location, direction and size into the current layer's redline information. The arrow's tip is first specified, followed by the arrow's tail which indicates the direction and length.



Add Line

This command allows you to add a series of straight lines into the current layer's redline information



Sketch

This command allows you to insert freeform drawing into the current layer's redline information. Simply clicking and dragging allows you to draw lines, curves and other shapes.



Delete

This command allows you to specify items you wish deleted from the redline information.



Properties

Active Layer

This command allows you to specify the d-code and color used when adding subsequent redline information. Existing d-code information is not affected by changing these properties.

Defaults

You may change the default line size and/or line color used. The specified line size will be mapped to an appropriate d-code. These settings are defaults only and may be overridden using the above settings.



View Redlining

This command allows you to specify whether you wish to see the associated redline information for all of the currently visible layers. When the item is checked the redlining information will appear on the screen as well as when printed.

Drawing

The Drawing menu provides sub menus containing commands that allow you to edit drawing information contained in your design. With these commands you can create quality mechanical drawings and diagrams complete with 'intelligent' drawing primitives which automatically update themselves.

Multiple drawings can be contained within the design by using the Layer Setup dialog and simply selecting the layer type 'drawing' for each layer you wish to have contain a drawing. Creating a drawing is a matter of ensuring that the desired drawing layer is the current layer and then using the appropriate functions found under this menu.

Drawing information, as with all other layer information, is stored in the design file. Drawings can also be plotted and exported in the same manner as other layers.

Chapter 7

The Drawing menu item presents a sub-menu with the following selections:

- Dimensioning
- Drill
- Fabrication

Dimensioning

The Dimensioning menu selection presents a sub menu of dimensioning commands. The menu selections include:

- | | |
|--------------------|---------------|
| ▪ Add Dimension | ▪ Add Line |
| ▪ Delete Dimension | ▪ Delete Line |
| ▪ Modify Dimension | ▪ Modify Line |
| ▪ Properties | |



Add Dimension

This command allows you to add one or more dimensions to the current drawing layer. Each dimension reports the distance between two features—i.e. flash center points, and draw endpoints—and/or text that you specify. After selecting OK in the Add Dimension dialog box you will be prompted for the features to dimension and for a point which will specify where to place the dimension text. If dimensioned items are moved, such as by editing or scaling operations you perform, the dimensions will automatically be updated. Dimensions automatically relocate dimension text and/or the direction of arrows as needed in the event that there is insufficient room to fit these between the dimension extension lines. This feature overrides text location and arrow direction settings that may have been specified in the Add Dimension dialog box.

When a dimension is created with the text box empty, the dimension text is automatically generated by taking the measured distance and appending the appropriate text specified in the Dimension Properties dialog (see Properties). If, however, the user specifies text in the text box it will be used to specify what to display as the dimension text. Two special sequences may be present in the text. The sequence '<>' (a less than and greater than symbol) specifies that the pattern is to be replaced with the distance, using current units. The sequence '[]' (an open bracket and closed bracket) indicates that the pattern is to be replaced with the distance in metric units if Imperial units are in use, or in Imperial units if metric units are currently in effect.

As an example, consider the following: If Imperial units are currently specified and you put the following into the text box...

‘This distance is <> (or [])’

Assuming the dimension is for features 10” apart and the Dimension Properties dialog contains the default settings, you would see the following resulting string...

‘This distance is 10” (or 254mm)’

Linear

Specifies that the dimension is to of two specified items.

Continued

Specifies that the first dimension will be of two specified items and that any subsequently specified dimensions will be relative to the last point so that all of the dimensions will appear as connected.

Baseline

Specifies that the first selected item will specify the starting point for all subsequent dimensions. Dimensions specified after the first one are automatically offset from the last dimension in the same direction as the dimension text (see Text Spacing).

Ordinate

Specifies that dimensions will be created in the same manner as Continued, only with no dimension lines between the extension lines and with the dimension text rotated 90 degrees.

Horizontal

Specifies that the new dimension will measure the horizontal distance between two features.

Vertical

Specifies that the new dimension will measure the vertical distance between two features.

Text Spacing

Specifies the distance to be used when spacing Baseline type dimensions (see Baseline).

Height, Width, Rotation, Line Spacing, Width, Slant, Char Spacing, Mirror, Text File

See the Add/Text command detailed eralier in this chapter.

Chapter 7

Left/Over

Specifies that dimension text is to appear to the left of the dimension (if this is a vertical dimension) or over the dimension (if this is a horizontal dimension).

Middle

Specifies that dimension text is to appear in the middle of the dimension (the default setting).

Right/Under

Specifies that dimension text is to appear to the right of the dimension (if this is a vertical dimension) or over the dimension (if this is a horizontal dimension).

Note: The Left/Over, Middle, and Right/Under settings may be overridden if dimension text and/or arrows will not fit between extension lines.

Options

This command allows you to specify other dimension line properties.

Arrow1 Type, Arrow2 Type

Specifies the type of arrows to use in the dimension.

Extension

Specifies the distance to offset the dimension extension lines from the dimensioned features.

Dimension

Specifies the line thickness to use when drawing the dimension.

Text

Specifies the line thickness to use when drawing dimension text.

Arrows Inside Extension Lines

When checked, specifies that arrows are to be drawn within the extension lines (the default case); otherwise, arrows are to appear outside of the extension lines. Note that this setting may be overridden if the dimension text and/or arrows cannot fit within the extension lines.

Box Around Text

When checked, specifies that a box is to be drawn around dimension text; otherwise, no box is to appear (the default case).



Delete Dimension

This command allows you to delete a dimension. After selection of this command you will be prompted to click the dimension you wish to delete.



Modify Dimension

This command allows you to change any property of a selected dimension. See Add Dimension for a description of dimension properties.

Properties

This command allows you to specify what text will appear immediately after a reported distance in the dimension text. The text would usually be used to indicate the units for the reported distance. Note that these are global settings that will automatically affect all dimensions in all drawings in the design where a distance is reported (see Add Dimension).

If Units are Imperial

If Imperial units are currently in use text appearing here will be appended to all dimension text. The default value is ‘ ” ’ (the double quote), indicating inches.

If Units are metric

If metric units are currently in use text appearing here will be appended to all dimension text. The default value is ‘mm’, indicating millimeters.



Add Line

This command allows you to add a construction line to the currently selected drawing layer. After selecting OK in the Add Construction Line dialog you will be prompted for the construction line beginning and ending points.

Type

Specifies the type of construction line to draw.

Width

Specifies the width of the construction line to draw.

Chapter 7



Delete Line

This command allows you to delete a construction line. After selection of this command you will be prompted to click the construction line you wish to delete.



Modify Line

This command allows you to change any property of a selected construction line. See Add Line above for a description of construction line properties.

Drill

The Drill menu selection presents a sub menu of drill drawing commands. The menu selections include:

- Add Hole Chart
- Delete Hole Chart
- Modify Hole Chart
- Update Hole Charts



Add Hole Chart

This command allows you to add a hole chart to the current drawing layer. Hole charts are intelligent and automate the entire process of drawing themselves, complete with tool information, automatic sizing of the chart, and assigning unique legend symbols to each drill tool used. Drill locations are automatically marked using the appropriate legend symbol. In addition hole charts can automatically redraw themselves, making it unnecessary to make manual modifications to hole charts when drill information has changed. Hole charts also automatically update themselves when the units have been changed.

To specify a title for a hole chart other than the default 'Hole Chart', enter the desired text into the provided text area.

Drill Layer

Specifies the drill layer that this hole chart will describe.

Height, Rotation, Line Spacing, Width, Slant, Char Spacing, Text File

See the Add/Text command detailed earlier in this chapter.



Delete Hole Chart

This command allows you to delete a hole chart. After selection of this command you will be prompted to click on the hole chart you wish to delete.



Modify Hole Chart

This command allows you to change any property of a selected hole chart. See Add Hole Chart for a description of hole chart properties.



Update Hole Charts

This command allows you to have all hole charts in the design redraw themselves, updating the information contained within them.

Fabrication

The Fabrication menu selection presents a sub menu of fabrication drawing commands. The menu selections include:

- Add Note Balloon
- Delete Note Balloon
- Modify Note Balloon



Add Note Balloon

This command allows you to add a note balloon to the current drawing layer. After selecting OK in the Add Note Balloon dialog you will be prompted for the placement of the note balloon and the endpoint of the leader line. If the leader line endpoint appears within the note balloon then no leader line will be drawn.

The text box contains the text that will appear within the note balloon.

Triangle

Specifies that the note balloon is to have a triangle appear around the text.

Circle

Specifies that the note balloon is to have a circle appear around the text.

Square

Specifies that the note balloon is to have a square appear around the text.

Chapter 7

Height, Width, Rotation, Slant

See Add/Text.

Line Spacing, Char Spacing, Text File

See Add/Text.



Delete Note Balloon

This command allows you to delete a note balloon. After selection of this command you will be prompted to click on the note balloon you wish to delete.



Modify Note Balloon

This command allows you to change any property of a selected note balloon. See Add Note Balloon for a description of note balloon properties.

Query Menu

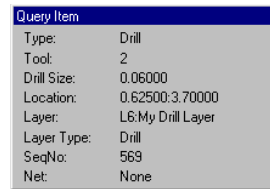
The main menu selection Query presents a menu of commands to provide information about the loaded database. The menu selections are shown below:

- Item
- Net
- UserData
- Highlight
- Measure
- Copper
- Extents



Item

This command allows you to obtain information on individual items within the database. As you cycle through the database selecting items, each selected item is highlighted and its item definition, along with its X-Y location and other information, are displayed in a form as shown below:



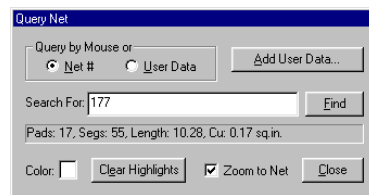
Query Item Display.

You may select items either by clicking directly on a database item or you may use the 'N', 'P' or 'G' hot keys to navigate to the Next, Previous or Goto directly to a particular item.



Net

This command allows you to highlight true multi-layer nets using a variety of colors. Selected nets remain highlighted until you specifically clear them.



Query Net dialog box

Query by Mouse, Net or UserData

You may select a net at anytime by pointing and clicking anywhere on a line segment or flash. You may also search for nets by their GerbTool net number or their UserData.

Search For

Enter the desired net or UserData value to search for.

Find

Click this button to have GerbTool find and highlight the net that contains the value in the Search For field.

Clear Highlights

Click this button to clear any nets currently highlighted. This does not change the net, only the highlighting is cleared.

Chapter 7

Color

You may change the color of subsequent net selections by selecting a new color. Previous selections are not altered.

Add UserData

Once you have a net selected you may globally add a UserData value to all items in the selected net. This provides an easy way of assigning meaningful net 'names' to your nets.

Zoom to Net

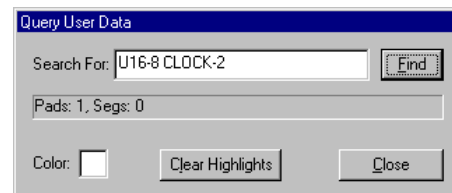
Check if you wish the view to change to display the entire selected net.

Note: This command relies on the netlist information supplied by a previous invocation of the Tools/Netlist/Generate command. If netlist information does not exist you will be prompted whether to create one.



UserData

This command allows you to highlight all items that contain a specific UserData value. Each selected item remains highlighted until you specifically clear them.



Query UserData dialog box

Search For

Enter the desired UserData value to search for.

Find

Click this button to have GerbTool find and highlight all items that contain the value in the Search For field.

Note: For a successful match, the search value can be located anywhere within an item's UserData value. This means that a search value of 'DATA' would match both 'DATA32' and 'MYDATA'. The comparison is also insensitive to upper/lower case.

Clear Highlights

Click this button to clear any items currently highlighted. This does not change any items, only the highlighting is cleared.

Color

You may change the color of subsequent selections by selecting a new color. Previous selections are not altered.

Measure

The Measure command presents a sub-menu with the following selections:

- Point to Point
- Edge to Edge
- Center to Center



Point to Point

Use this command to obtain accurate measurements of your data. GerbTool first prompts for a base point to measure from. As you move the cursor away from the base point the distance in X and Y as well as true length will be displayed in the prompt area. A left click will change the base point to the current cursor position.



Edge to Edge

This command measures the actual minimum distance between two Gerber data items. GerbTool first prompts you to select a base item. As you select additional items, the actual minimum distance between items in X and Y as well as true length will be displayed in the prompt area.



Center to Center

This command measures the actual distance between the centers of two Gerber data items. GerbTool first prompts you to select a base item. As you select additional items, the actual distance between the center of the items in X and Y as well as true length will be displayed in the prompt area.



Highlight

Use this command to highlight all occurrences of a specified D-Code. You may restrict your selection to flashes, draws or both and a particular

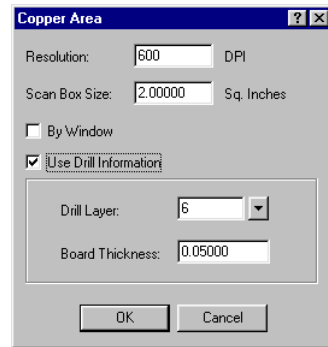
Chapter 7

layer. The selected D-Codes remain highlighted until you turn off the highlight with the View/Highlights command, 'H' nested command or you select another group of items with this command.



Copper

This command will accurately calculate the amount of copper used on one or more layers using a high resolution bitmap method. All visible layers will be scanned.



Copper Area Dialog.

Resolution

Specify the bitmap resolution to use in calculating the copper area in dots per inch (DPI).

Scan Box Size

Specify the size of the bitmap to use during the scanning process.

By Window

Check if you wish to calculate the copper area of a user-specified window.

Use Drill Information

Check to use drill information for more precise calculations. Hole diameters are subtracted from pad sizes and hole barrel calculations are performed.

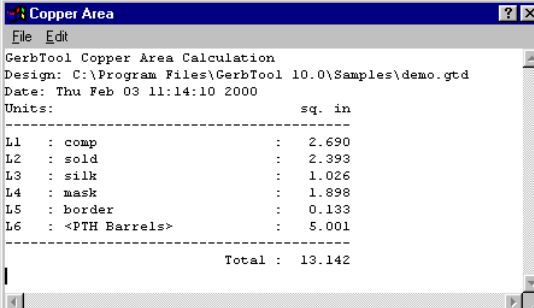
Drill Layer

Specify the drill layer to use when making calculations.

Board Thickness

Enter the board thickness here. This is used when calculating hole barrels.

After entering in the information on the copper area dialog and selecting OK, the copper area will be calculated and the following report will be displayed.



Layer	Description	Area (sq. in)
L1	comp	2.690
L2	sold	2.393
L3	silk	1.026
L4	mask	1.898
L5	border	0.133
L6	<PTH Barrels>	5.001
Total		13.142

Copper Area Report.

You may edit the report, copy it to the clipboard, save it to a file or print it on a printer using the pull down menus provided.



Extents

Use this command to determine the data extents of all layers loaded. In addition to displaying the extents information, GerbTool also updates its internal data extent information. This will allow the View/All command to correctly center the data after you've made edits to the database. By selecting the True Size toggle button, you control whether the extents displayed takes the true size of each database item into account or just their center points. Selecting the Include Virtual Panelization toggle button will allow virtual panels to be included in the extents calculations also.

Options Menu

The Options menu selection displays a menu of commands that control the GerbTool environment. The menu selections include:

- Grid Snap
- Ortho Line Snap
- Arcs 360
- Units and Precision
- Configure



Grid Snap

Use this command to toggle grid snapping on/off. See the Configure command below for information on changing the appearance of the grid.

Note: This command is also available as a nested command by touching the Ctrl+S key (see Chapter 4, 'Nested Commands' for more information about nested commands).



Ortho Line Snap

This command allows you to toggle orthogonal snap mode on/off. When enabled, all lines drawn interactively will be forced to the specified angle. See the Configure command below for information on changing the snap angle.

Note: The current setting may be temporarily overridden by holding down the Ctrl key.



Arcs 360

This command allows you to toggle interpolated arcs on/off. This setting affects the method of creating arcs used by the Add/Arc Ctr, Add/Arc 3 Pt and Add/Circle commands. If enabled all arcs will be created using 360° circular interpolation. If disabled, all arcs will be created using small line segments. This does NOT affect the way Gerber data is read from a disk file. It only pertains to adding new arcs with the above mentioned commands.

Note: Not all photo-plotters support circular interpolation.



Units and Precision

This menu item serves as a shortcut to the Options/Configure/Units and Precision dialog explained later in this chapter.

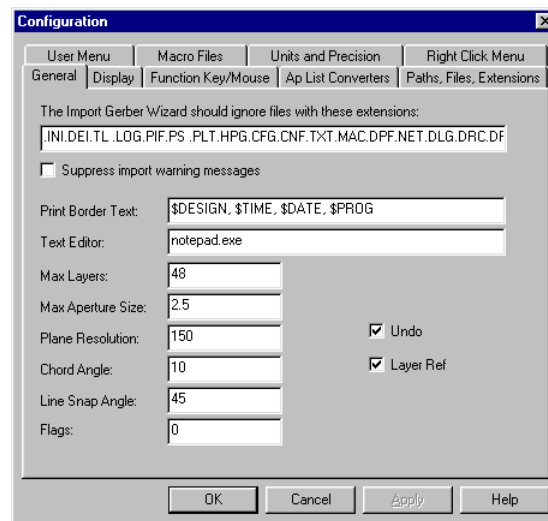
Configure

This command allows you to change the way GerbTool is currently configured and many of the default settings that GerbTool uses at startup. A tabbed dialog box is displayed that contains the following topics:

- General
- Display
- Function Key/Mouse
- Ap Converters
- Paths, Files, Extensions
- Macro Files
- Units and Precision
- User Menu
- Right Click Menu

General

This tab displays a form showing the current values of various general program settings.



General Program Settings.

Chapter 7

The Import Gerber Wizard should ignore files with these extensions

Enter any filename extensions that you know are not used for Gerber or aperture list files. The more extensions that GerbTool is able to ignore the faster the Import Gerber Wizard will be able to build a file list by not wasting it's time on files known not to be valid Gerber or aperture list files.

Print Border Text

This field allows you to specify what text will appear in the border of check plots generated when printing and exporting to HPGL and PostScript. GerbTool looks for the key words \$DATE, \$TIME, \$DESIGN and \$PROG. If any of these keywords are found they will be replaced with the appropriate text. All other text specified will be included in the border verbatim.

Suppress import warning messages

Check to prevent warning messages from being displayed when importing files. A log file will still be created but you will not be prompted whether to view it.

Text Editor

This field allows you to specify what text editor GerbTool will invoke when you are presented with a file to view or edit.

Max Layers

This field allows you to control the number of layers that GerbTool may handle. The valid range of values is 32-999. Use the minimum value that satisfies your requirements to conserve memory.

Note: This parameter will not become effective until the next time you start GerbTool.

Max Aperture Size

This field specifies the maximum aperture size that GerbTool will create when creating filled polygons.

Plane Resolution

This field allows you to specify the "Dots Per Inch" resolution of the bitmap created when processing a pwr/gnd plane during netlist generation. To allow maximum speed, keep this value to a minimum. Default is 150 DPI.

Chord Angle

This field allows you to specify the chord angle used when creating segmented arcs using editing commands. For example, a chord angle of 5° would result in a 18 separate line segments for a 90° arc.

Line Snap Angle

This field specifies the angle at which lines will be forced to if Ortho Line Snap is enabled.

Flags

This field allows you to control some aspects of GerbTool's low level operations in the field. Typically you would be instructed by GerbTool Technical Support personnel on how to modify this parameter. The value is entered as a hexadecimal number.

Undo

This check box specifies whether the saving of undo information is on or off and whether it should be initially on upon startup. If undo is currently enabled, and you disable it, any current undo information will be destroyed and undo will then be disabled.

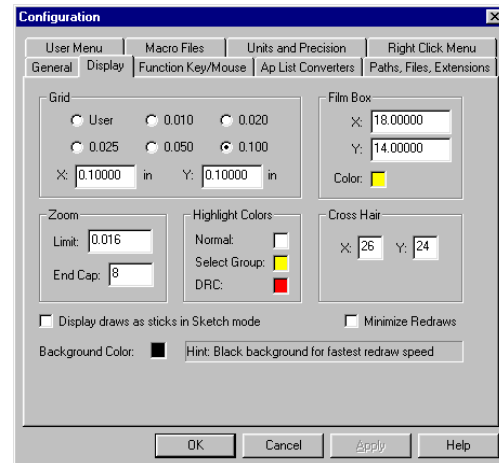
Layer Ref

This check box controls whether 'ref' layer status is available. If you find that you don't use ref status you may disable it by un-checking this box.

Chapter 7

Display

This tab displays a form showing the current settings that affect the GerbTool desktop.



Display Parameters.

Grid

You may select a pre-defined grid size or, by entering a value in the size X/Y fields, you may specify a non-standard grid size.

Film Box

You may change the current film box size, by editing the size X/Y fields, and/or the film box color by clicking on the Color button.

Limit (Zoom)

This field allows you to limit how far GerbTool can zoom in. On certain combinations of screen resolution and file format the display of items at extreme magnification can appear distorted. This setting allows you to prevent this from occurring.

End Cap (Zoom)

This field specifies when GerbTool should stop attempting to draw end caps on drawn lines. If the thickness of a line (in pixels) is less than or equal to this parameter, no end caps will be drawn. Higher values provide decreased redraw times at minimum zoom levels.

Note: This parameter only affects redraw speed and has no effect on your database.

Highlight Colors

These color buttons allow you to control the colors used when highlighting database items.

Cross Hair

The X and Y fields provide control over the size of the drawing area cross hair cursor. Enter 0,0 for a full screen cursor.

Display draws as sticks in Sketch mode.

This button toggles whether sketch mode displays draws as outlines or single line “sticks”.

Background Color

This button provides the ability to change the Drawing Area background color. As with all color buttons within GerbTool, simply click on the color button for a list of available colors.

Hint: For fastest redraw speed specify a black background and use Overlay(Translucent) display mode.

Minimize Redraws

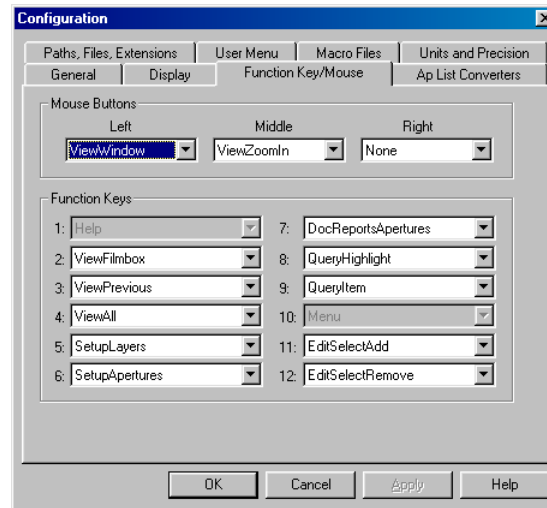
This check box controls whether GerbTool minimizes the amount of redrawing it does on your behalf. If you are an experienced user you may be comfortable redrawing the screen only when you want to.

Note: Regardless of this setting, you may always interrupt a redraw, without affecting the current command, by pressing the [Esc] key.

Chapter 7

Function Key / Mouse

This tab displays a form showing the current function key/mouse command assignments.



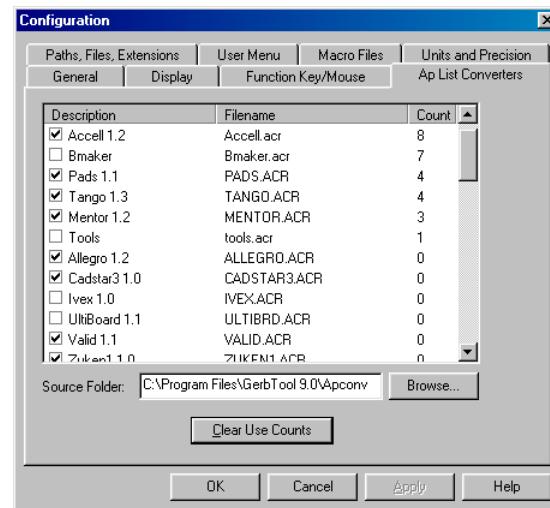
Function Key/Mouse Assignment Form.

You may change any of the commands assigned to the mouse and function keys by selecting from the drop down lists. Any changes you make will become effective immediately after clicking on the OK button. This will also save the current key assignments so they will be available the next time you start GerbTool

Note: In addition to command names, function keys may also be programmed with GerbTool Macros allowing virtually all of GerbTool's power to be within one keystroke.

Ap List Converters

This tab displays a list form where you select which aperture list converters you require.



Aperture List Converters.

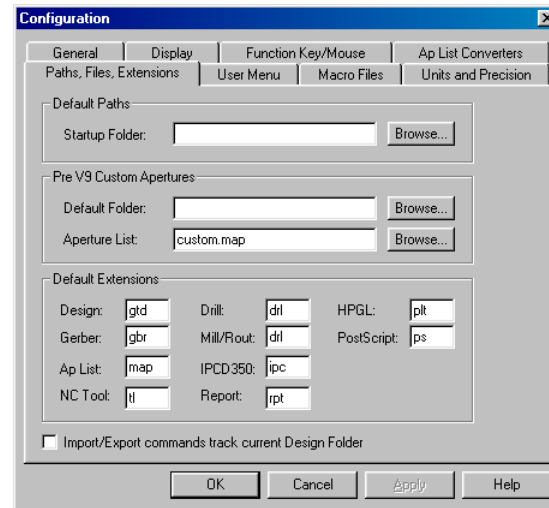
This list shows all aperture list converters found in the GerbTool program folder. Converters that are currently selected will be at the top of the list with a check mark by them. You may select or de-select converters as required. A rule of thumb is less is better. You may also change the title of a converter by clicking on an item in the list and then clicking on the title. You may then type in a new name. The use counts associated with each converter allows GerbTool to try the most popular converters first during automatic aperture list conversion. This can speed up the process considerably.

Note: To add additional ACR files to GerbTool, simply copy them into the folder specified by the Source Folder field.

Chapter 7

Paths, Files, Extensions

This tab displays a form containing edit fields for various program default values regarding paths, files and extensions.



Paths, Files and Extensions.

Startup Folder (Default Paths)

Fill in this field if you want GerbTool to switch to this folder immediately after startup. If this field is blank, GerbTool will switch the folder you were in when you exited GerbTool last.

Default Folder (Pre V9 Custom Apertures)

Fill in this field if you want GerbTool to always look for old style custom aperture files (.cus) in this directory. This allows you to keep all your custom apertures in one location if desired.

Aperture List

This field specifies the aperture list used by all old style custom aperture files (.cus) loaded. An aperture list used for custom apertures should not itself contain any custom apertures.

Default Extensions

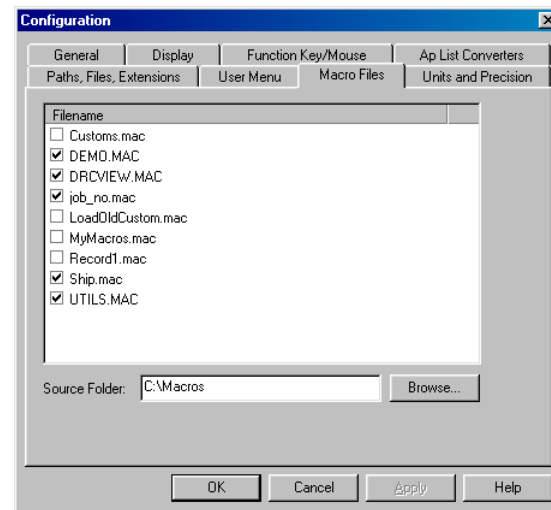
This group of edit fields specify the default filename extensions for the indicated file types.

Import/Export commands track current Design folder

This check button indicates whether GerbTool should remember the folders where you last imported/exported files. If checked, GerbTool will always assume that your next import/export will be from/to the current design folder. You may always navigate to another folder, but the file chooser will default to the current design folder.

Macro Files

This tab displays a list where you select which Macro files you want to load the next time you start GerbTool.



Macro Files.

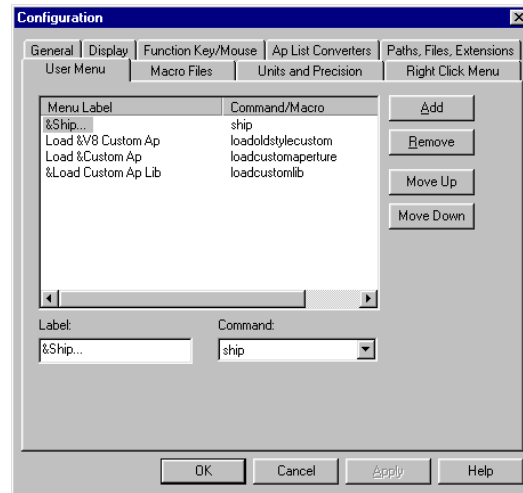
This list shows all Macro files found in the GerbTool program folder. You may select or de-select Macro files as required.

Note: To add additional Macro files to GerbTool, simply copy them into the folder specified by the Source Folder field.

Chapter 7

User Menu

This tab displays the current user menu configuration and allows you to make changes as desired.



User Menu setup.

Label

This field allows you to type the menu item label text. A character prefixed with the '&' character will be considered the menu item hot key.

Command

This dropdown list allows you to select either a Macro name or command name.

Add

Click this button to add a new item to the User Menu. The current value in the Label and Command fields will be used to construct the menu item.

Remove

Click on an existing menu item in the list and then click this button to remove the item. If you add an item and then want to change it, you must remove it and re-add it.

Move Up

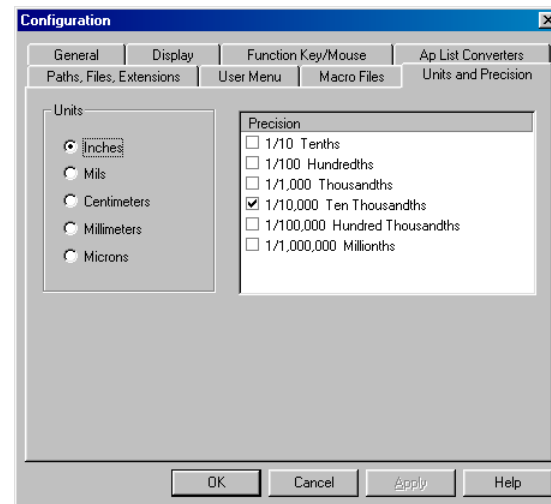
Click this button to move the currently selected list item up one position.

Move Down

Click this button to move the currently selected list item down one position.

Units and

This tab displays the current units and precision settings and allows you to make changes as desired. Precision



Units and Precision setup.

Use this dialog to control how information within GerbTool that represents sizes and distances (i.e. coordinates) will be presented.

Warning: There is a possibility of rounding errors within dialogs that can occur when switching back and forth between different settings. Use care when making such changes.

Units

Select the type of units.

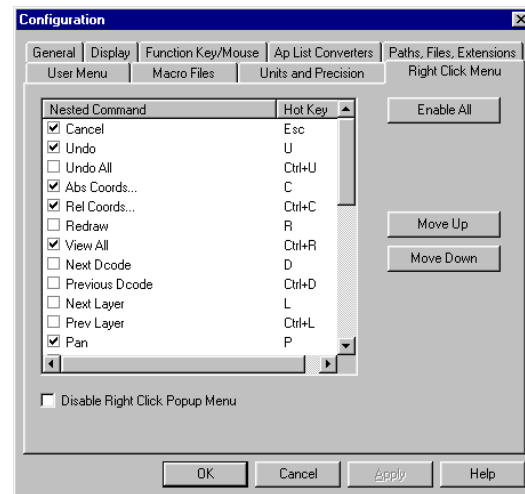
Precision

Select the desired precision.

Chapter 7

Right Click Menu

This tab displays the current right click menu configuration and allows you to make changes as desired. Right clicking your mouse within the drawing area pops up a context sensitive menu of available nested commands. You may control what commands are listed and in what order using this dialog. See Chapter 4, 'Nested Commands', for more information on what nested commands do and how to use them.



Right Click Menu setup.

Put a check mark by the commands that you would like to include in the right click menu. Commands without a check mark will still be available using the indicated 'hot key', but will not be included in the right click menu.

Enable All/Disable All

This button allows you to toggle all commands on or off.

Disable Right Click Popup Menu

Check if you wish not to have the right click menu appear when clicking the right mouse button.

Move Up

Click this button to move the currently selected list item up one position.

Move Down

Click this button to move the currently selected list item down one position.

Macro

The Macro main menu selection presents a menu of commands for dealing with GerbTool Macros. A complete description of GerbTool Macros and the powerful Macro Developer is provided in the online help available in the Macro Developer.

- Run
- Load
- Developer
- Record

Run

This command will prompt you to select a macro to run. All macros loaded at program startup and through the Macro/Load or Macro/Record commands will be available for execution.

Load

Use this command to load additional macro files into GerbTool. This allows any macros present in the specified file to be included in GerbTool's list of available macros.

Note: To have GerbTool automatically load a Macro file at startup, see the Options/Configure command described earlier in this chapter.

Developer

This command starts the Macro Developer. If the Macro Developer appears minimized on the screen this selection will restore the Macro Developer window's size, allowing you to edit the macro currently contained within it. Otherwise, if the Macro Developer is not already on the screen, you will be prompted to select a macro file to edit. The selected macro file will then be loaded into the Macro Developer, if it already exists, and displayed on the screen.

Chapter 7

Record

This command toggles the Macro Developer's record mode on and off. When selected the Macro Developer window will appear minimized if it is not already displayed on the screen. Both the Macro Developer title bar and the presence or absence of a checkmark on this item indicate whether the record mode is currently on. When the record mode is on user commands are automatically captured and recorded to the Macro Developer. Once the desired sequence of commands has been recorded, the recording can be stopped by simply selecting Macro/Record again to toggle the selection off.

To save the recorded macro:

- ✓ Open the Macro Developer by either selecting Macro/Edit or by clicking on the Restore or Maximize buttons on the minimized Macro Developer window.
- ✓ If desired, change the name of the macro on the MACRO line to reflect what the newly created macro does.
- ✓ Select File/Save from the Macro Developer's menu.

To run the macro, select Debug/Run Macro in the Macro Developer or, within the GerbTool menu, select Macro/Load and then Macro/Run.

Tools Menu

The Tools menu selection will display a menu of commands that provide important CAM capabilities. Some commands will operate on a window or groups of items as well as complete layers. The menu selections include:

- | | | |
|-------------|------------------|-------------------|
| ▪ Panelize | ▪ Fix SilkScreen | ▪ Convert |
| ▪ Netlist | ▪ Pad Removal | ▪ Layer Spread |
| ▪ DRC | ▪ Drill | ▪ Vent/Thieving |
| ▪ Snoman | ▪ Mill/Rout | ▪ Compare Layers |
| ▪ Teardrops | ▪ Test Points | ▪ Find Duplicates |



Panelize

The Panelize command is used to create multiple copies of a design. This allows multiple copies of the design to be manufactured as one panel.

Panelize Edit Form.

Automatic Panelization

All that's required to panelize an image is to locate the desired data within the film box (see Edit/Origin), view the layers that are to be panelized and enter the desired image border to border spacing in the X & Y fields of the Edge to Edge spacing group box within the Panelize edit form.

Note: Although only visible layers will be copied, all layers of the original image will remain aligned after panelization.

Manual Panelization

To panelize **manually**, remove the check mark from the Auto check button if needed. You must also enter the number of rows and columns in the appropriate fields as well as the Point to Point distance between copies.

Chapter 7

Automatic Venting

Automatic Venting occurs during panelization, whether automatic or not. To vent a panel automatically, check the Auto Vent button within the Panelize edit form. You may also define the spacing between the image data extents and the venting area with the Vent/Image Spacing field, specify the D-Code and spacing between the flashes in the vent pattern and the layer to add the vent pattern to.

Note: In both automatic and manual venting, the style of vent pattern is easily customized using custom apertures. For example, you could create a hatch or cross-hatch pattern using a diagonal or cross shape custom aperture. Just be sure to set the height and width of the overall size of the custom aperture in the aperture list.

Virtual Panelization

Enabling the Virtual button within the Panelize edit form allows GerbTool to panelize your design without actually duplicating layer data.

Note: Although no data is duplicated during virtual panelization, the data origin is modified to center the images within the panel. Therefore, it is still necessary to save your design after panelization.

Virtual panelization provides many benefits including automatic updating of all images during edits and drastically reduced file sizes. Furthermore, if your designs are to be plotted on a 274-X, FIRE9XXX or EIE compatible plotter, GerbTool will automatically insert the proper step & repeat codes into your Gerber data.

Note: If your designs are to be plotted on a plotter that does not support step & repeat codes, you must execute the Panelize command without the Virtual button enabled and export your panelized Gerbers before you actually send them to the plotter.

GerbTool will also insert step & repeat codes into NC Drill and Mill/Rout output data if the Virtual button is enabled. This may be necessary to drill large panels if your NC equipment is memory limited.

Virtual panel mode and hence the display of virtual panels may be toggled on/off using the 'Ctrl+V' nested command. See Chapter 4, 'Nested Commands' for more information about nested commands.

Clicking on the Virtual Layers button will popup a list of loaded layers from which you may choose which layers are to be included during virtual panelization.

Note: You may right click or touch the escape key to stop the drawing process anytime during the panelizing process. This usually provides a noticeable improvement in the overall time to complete the panelizing process without effecting the finished panel in any way.

Netlist

The Netlist command presents a sub-menu of commands as shown below:

- Generate
- Save



Generate

The Generate command will process all viewed conductive layers and create a single multi-layer netlist that becomes part of the internal database. The netlist may then be used by other commands that require a netlist such as IPC-D-356 import and Design Rule Checking.

Don't include items in Select Group

This setting allows you to ignore all items that are in the current select group. For example, if you have blocks of text that you know you don't want included in your netlist, you may use the Edit/Select/Add command to create a select group with these blocks of text. Netlist generation will then ignore the text thereby producing a much cleaner netlist.

Thru-hole Drill Layer

Specify your drill layer here. GerbTool uses your drill layer to determine layer to layer connectivity as well as direct connections to plane layers. If you do not supply a drill layer, GerbTool will use the aperture list type field to determine whether each pad is SMT or not.

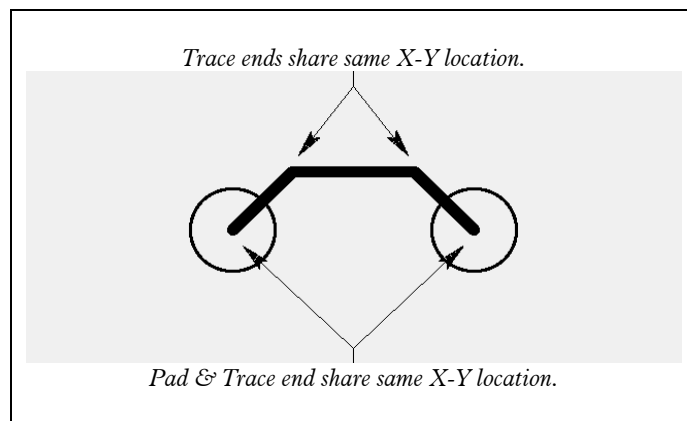
Setup Blind/Buried Via Layer Sets

Clicking this button pops up a dialog that allows you to define Blind/Buried via layer sets.

Chapter 7

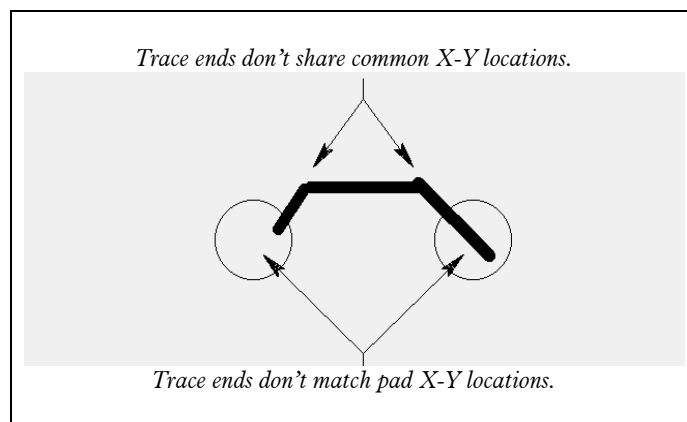
Well Behaved

This setting allows you to indicate whether your database is 'Well Behaved' or not. A well behaved Gerber file is defined as one where all items that are to be considered connected share a common X-Y location as shown below:



Example of a 'Well Behaved' Gerber file.

The following illustration shows an example of a Gerber file that is NOT well behaved:



Example of a Gerber file that is NOT 'Well Behaved'.

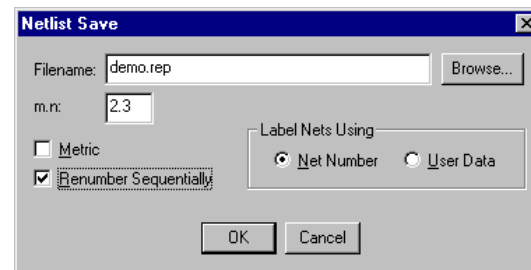
If you determine that your Gerber files are indeed well behaved, it is recommended that you choose this mode when generating a netlist as there is a dramatic increase in processing speed due to the well behaved nature of the Gerber files.

Since so many CAM tools require a netlist to perform properly, you may save the generated netlist within your Gerber files for later use. If netlist saving is enabled using the Gerber Export Format dialog, and a netlist is present, it will be saved when the layer is exported to disk. To remove a netlist from a Gerber file, simply import the layer (or layers) and disable netlist saving using the Gerber Export Format dialog when exporting the necessary layers.

Note: GerbTool uses the 'G04' command to embed a netlist within a Gerber file. This will cause the Gerber file to increase slightly in size. If your photoplotting equipment is known to have problems handling the Gerber G04 command it is recommended that netlists be removed as described above before submitting your files to be photoplotted.

Save

This command will generate an ASCII netlist file consisting of net numbers, or names using UserData, and pad X-Y coordinates. See APPENDIX C, 'Sample Netlist File'. This command uses all viewed layers in generating the netlist file.



Netlist Edit Form.

Filename

Specify the desired output filename or use the Browse button to locate the desired output file.

M.N

Specify the coordinate format.

Chapter 7

Renumber Sequentially

This option instructs GerbTool to renumber the net numbers if needed to make sure that they are output in order and with no gaps in the net numbers.

Label Net Using

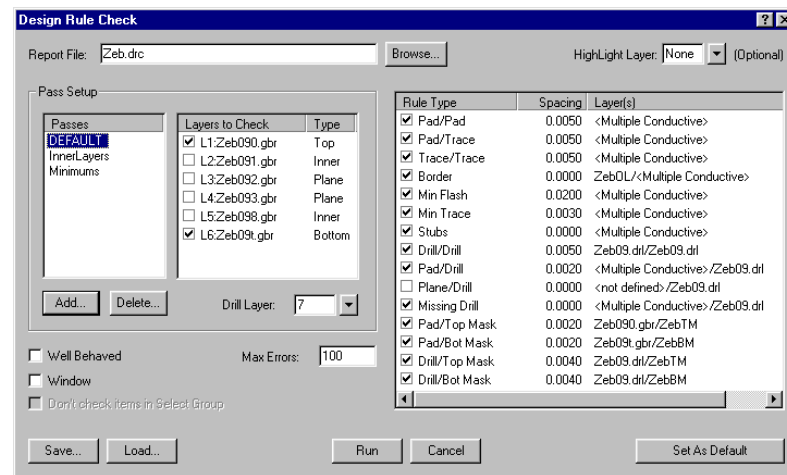
You have a choice of how to label the nets. If your nets have UserData assigned to them you may choose to have your netlist labeled with the UserData instead of net numbers.

Note: If a valid netlist does not already exist you will be prompted whether to generate one now. A netlist is required for this command to work properly.



DRC

Selecting this menu item invokes the DRC tool. The DRC tool is available to verify that your design meets the design rules that you specify. You may setup multiple passes that will be executable sequentially when you click the Run button. This feature allows more precise control of the DRC rules that are applied to your design. Each "pass" defines a set of layers and the design rules that apply to them. You can define as many "passes" as needed to fully check your design in one DRC run. You may also save/load rule sets to external files to create templates that can be used on similar designs.



DRC Setup Dialog.

Report File

This is the file that all errors will be logged to.

Passes

You may Add/Delete passes as needed using the list and action buttons. You may also right click a list item to change the name of an existing pass.

Layers to Check

Select the layers that the specified rules will be check against.

Drill Layer

Specify the drill layer used in the annular ring check here. A '0' also disables annular ring checking.

Highlight Layer

This field allows you to specify an optional layer in which all database items that are part of a DRC error will be copied. This allows you to easily see the whole picture and to make printing all errors at once possible. To disable this feature enter 'none' or '0' in this field.

Pad/Pad

Minimum spacing allowed between pads.

Pad/Trace

Minimum spacing allowed between pads and traces.

Trace/Trace

Minimum spacing allowed between traces.

Border

Minimum spacing allowed between any item and the border specified in the Border Layer field.

Min Flash

Minimum pad size allowed.

Min Trace

Minimum trace size allowed.

Chapter 7

Stubs

This will allow GerbTool to locate and highlight all trace stubs. A trace stub is defined as any trace that touches a pad or trace on one end, but does not on the opposite end.

Pad/Drill

Specify the minimum Annular ring required between conductive layers and drill layer here. The annular ring check compares the DRC layer(s) to the drill layer.

Plane/Drill

Specify the minimum Annular ring required between plane clearances and drill sizes here. The annular ring check compares plane layers to the drill layer.

Drill/Drill

Specify the minimum spacing required between adjacent drill locations.

Missing Drill

Pads that do not have a corresponding drill will be reported as a 'Missing Drill'.

Pad/Top Mask

Specify the minimum Annular ring required between top side pads and top soldermask here.

Pad/Bot Mask

Specify the minimum Annular ring required between bottom side pads and bottom soldermask here.

Drill/Top Mask

Specify the minimum Annular ring required between drill layer and top soldermask here.

Drill/Bot Mask

Specify the minimum Annular ring required between drill layer and bottom soldermask here.

Well Behaved

The DRC command supports two separate modes; 'Well Behaved' and normal. In the well behaved mode, GerbTool assumes that legal pad/trace or trace/trace connections will have common X-Y locations (see Chapter 7, 'Tools/Netlist' for a description of well behaved Gerber files). This means that ANY actual contact between items that don't share a common X-Y location, and are in different nets, will be

considered a violation. Conversely, in normal mode, any actual contact between items will not be considered a violation. Only items that are not in contact but are within the minimum spacing rules will be considered in violation. The well behaved mode is preferred if your Gerber files were produced accordingly as it provides much faster processing and more accurate results.

Note: If a valid netlist does not already exist you will be prompted whether to generate one now. While a netlist is not a prerequisite to DRC, a netlist increases the usefulness and correctness of DRC.

Window

You may optionally select window mode to run DRC on just a window of data versus the complete layer.

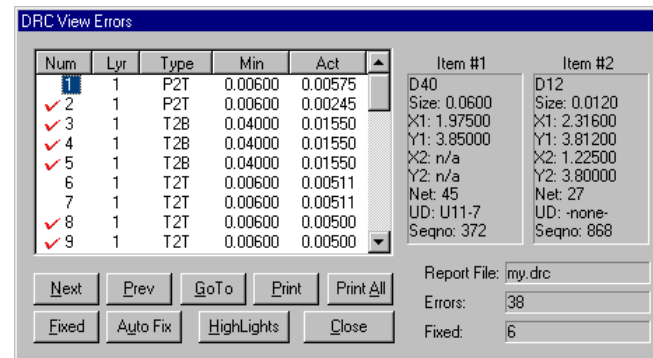
Max Errors

This field allows you to set a limit of how many errors DRC will produce. This prevents DRC from generating a possibly huge report file if you enter the incorrect spacing rules for a given design.

Don't check items in select group

This check button allows you to tell DRC to ignore all items that are in the current select group. For example, if you have blocks of text that you know you don't want DRC to check, you may use the Edit/Select/Add command to create a select group with these blocks of text. DRC will then ignore the text thereby producing a much cleaner report.

After performing the DRC, GerbTool will then enable DRC error viewing and display the DRC View Errors dialog box as show below:



DRC View Errors.

Chapter 7

You may use this dialog box to examine and document any or all DRC errors found.

Next

Jump forward to the next non-fixed error.

Prev

Jump back to the next non-fixed error.

GoTo

Jump to the current error, fixed or not.

Print

Print the currently selected error on the default Windows printer. The error report includes a screen capture of the error and all related information about the error and items involved.

Print All

Prints a separate error report for each error in the list.

Highlights

This button toggles the display of the current error highlights. Use this button to temporarily turn off highlights to allow easy correction of the error.

Fixed

Toggles the status of an error. Setting the status to fixed allows you to tell at a glance what errors have already been corrected.

Auto Fix

Clicking on this button executes the DRC Auto Fix command. GerbTool will calculate a suggested correction for the selected error, display it, and then prompt you to confirm it. If you agree with the suggested correction and click the Yes button, GerbTool will implement the correction.

GerbTool implements DRC corrections using primarily negative data to create the required clearances between features. This negative data is placed on a separate layer with a layer polarity of "CLEAR".

A unique situation occurs when a pad and a trace not only violate a spacing rule but actually overlap. In this case, the offending trace must be re-ordered to follow the negative correction data. An additional positive, or "DARK", layer is created to accomplish this. The end result is properly "shaved" pads.

When making a correction and you are prompted with the confirming dialog box, you have a couple of options. You may choose Yes, at which time the correction will be made as described above. Or, you may decline by clicking the No button. Or, you may click the “Fix All Errors” button allowing GerbTool to correct all remaining errors in batch mode. Or finally, you may click the Options button. This will popup a dialog box where you may choose whether you would like any necessary composite layers created without prompts and whether to ignore trace to trace errors during batch mode.

The following is a list of important points to remember when using the DRC Auto Fix tool:

- By nature, only RS274X and FIRE9000 file formats support negative data.
- Since GerbTool modifies the layer setup by adding additional composite layers, and subsequently updates the DRC report file to reflect these changes, the design file and corresponding layer data must be saved to keep everything in sync. If they are not saved, the DRC command will have to be re-run to restore the DRC report file.
- As DRC corrections make use of composites layers, composite viewing must be enabled to view the corrections properly. While running this command enables this mode automatically, you may use the View/Composites command to toggle this mode on and off.
- **Caution:** While the automatic correction of DRC errors can be a real timesaver, all automatic corrections made by GerbTool, especially those made in batch mode, should be carefully reviewed. While most corrections do not themselves cause problems, there may be complex error situations that cannot be correctly fixed automatically. Also, please note that further DRC runs will not take into account the corrections made due to their graphical nature.



Use the View/Errors command to toggle the display of the DRC View Errors dialog box.

Note: GerbTool updates the DRC report file generated by the last run of the DRC command with the fixed status of each error. Therefore, you should not remove or substantially alter this file if you intend to view DRC errors again using this file.

Chapter 7



Snoman

This menu selection will invoke the Snoman tool. Snoman is a highly configurable form of the method of eliminating pad/trace separation that is often referred to as filleting or teardropping (see APPENDIX B for a technical description of Snoman). The purpose of Snoman is to increase your manufacturing yield by adding more copper in the area of the pad/trace junction thereby eliminating any possible pad/trace separation. Snoman is used primarily when dealing with very small pads and traces such as micro vias in the 30 mil or less range but can be used anywhere to prevent pad/trace separation. Snoman provides additional versatility by allowing user control of the size and location of the generated Snoman pads along with an integral DRC to eliminate any possible spacing violations.

Trivia: Snoman derives its unusual name from the appearance of a Snoman pad placed on-top of a host pad which resembles a real ‘Snowman’.

The Snoman tool will create a *maximum material condition* at the point of trace entry into a pad.

Snoman Edit Form.

Enter a valid filename in the Report File field as any errors will be logged to this file. You must specify a layer to operate on (From Layer) as well as an output layer (To Layer) for the generated Snoman pads.

Note: If you enter a ‘0’ in the From Layer field, all viewed layers will be processed with the resultant Snoman pads being added to their respective layers.

You may restrict the generation of Snoman pads to a particular D-Code by entering a D-Code in the D-Code field. A D-Code of '0' matches all. Edit the spacing parameters to specify the design rules that Snoman must adhere to. The Host Offset field contains the offset maintained between the host pad centroid and the edge of the generated Snoman pad. This value may be negative. If Snoman detects a spacing rule violation while placing a Snoman pad, it will reduce the size of the Snoman pad to avoid such errors. You may control to what percentage of the host pad size that Snoman may reduce the size of the Snoman pad. Use the Min Percent field to specify this value. The Max Percent field allows you to control the maximum size of the generated Snoman pad as a percentage of the host pad size. You may also indicate whether Snoman should operate on a window of data versus a complete layer.

Note: If a valid netlist does not already exist you will be prompted whether to generate one now. A netlist is required for the Snoman tool to work properly.

Use the View/Highlights command to view 'would be' rule violation errors, if any, after executing this command.



Teardrops

This menu selection will invoke the Teardrop tool. The Teardrop tool will create a *maximum material condition* at the point of trace entry into a pad.

The Teardrops dialog box contains the following fields and controls:

- Report File:** A text field containing 'demo.txt' and a 'Browse...' button.
- Layer:** A dropdown menu showing '2'.
- D-Code:** A dropdown menu showing '0'.
- Pads:** A section containing a 'Percent of Host' field set to '15'.
- T-Junctions:** A section containing 'Length Multiple' and 'Width Multiple' fields, both set to '0.5'.
- Minimum Spacing:** A section containing 'Pad/Trace' and 'Trace/Trace' fields, both set to '0.006'.
- Window:** A checkbox that is currently unchecked.
- Delete Existing Teardrops:** A checkbox that is currently unchecked.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom.

Teardrop Form.

Chapter 7

Enter a valid filename in the Report File field as any errors will be logged to this file.

Note: If you enter a '0' in the Layer field, all viewed layers will be processed with the resultant Teardrops being added to their respective layers.

You may restrict the generation of Teardrops to a particular D-Code by entering a D-Code in the D-Code field. A D-Code of '0' matches all. Edit the spacing parameters to specify the design rules that Teardrop must adhere to. Pad tail length is calculated as a percentage of the host pad diameter. This value may be less than or greater than 100% and is specified in the Percent of Host field. "T" junction tail length and width is calculated as multiples of the host trace diameter. These values may be fractional and are specified in the Length Multiple and Width Multiple fields. You may also indicate whether Teardrops should operate on a window of data versus a complete layer. Choosing the Delete Existing Teardrops option allows the removal of existing Teardrops if any.

Note: If a valid netlist does not already exist you will be prompted whether to generate one now. A netlist is required for the Teardrop tool to work properly.

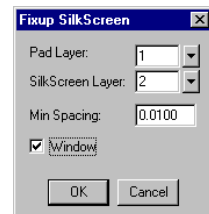
All pad locations that a Teardrop could not be generated for will be highlighted and their locations will be specified in the generated report file.



Fix SilkScreen

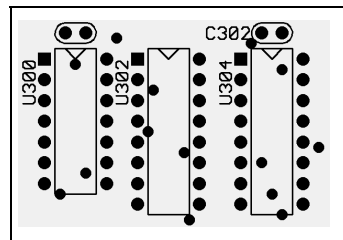
This command will automatically clean up silkscreen data around pads.

GerbTool will then 'clean-up' all places where silkscreen lines are to close to a pad.

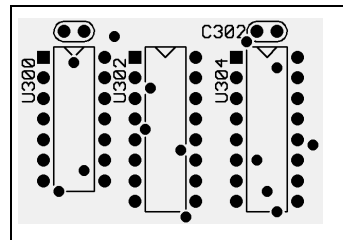


Fix Silkscreen edit form.

You specify the layer that contains the pads (Pad Layer) and the layer that contains the silkscreen data (SilkScreen Layer) as well as a minimum spacing to be maintained. If desired, you can use window mode to clean-up isolated areas rather than the entire silkscreen layer. GerbTool will then automatically adjust lines segments that violate the minimum spacing requirement. Each offending line is moved just enough to eliminate the violation and no more as shown in the following before and after sequence:



Before running the *Tools/Fix SilkScreen* command.



After running the *Tools/Fix SilkScreen* command.

Pad Removal

This command will present a sub-menu of commands as shown below:

- Isolated
- Stacked



Isolated

Selecting this command will remove any unused pads (isolated/floating pads) from inner layers.

Note: Only layers with a layer type of Inner will be considered. Use the Setup/Layers command to change this if necessary.

Chapter 7

GerbTool does not remove targets and/or thermal pads. You specify the layer to remove the pads from and whether you desire window mode versus processing the entire layer.

Stacked

Selecting this command will remove any unnecessary pads that are identical and stacked one on top of another, or are completely buried, and on the same layer. You specify the layer to remove the pads from and whether you desire window mode versus processing the entire layer.

Drill

The Drill menu provides commands that allow you to rearrange the ordering of information contained within drill layers in your design.

Unless otherwise specified, exported drill data will appear in the same order as the data originally appeared in imported drill files or manually inputted data. Reordering drill data is accomplished by either explicitly changing the drill sequence number associated with the desired drill items or by optimizing the data using the Optimize command.

Items with a higher drill sequence number are drilled later than those with a smaller sequence number. An item's drill sequence number is not necessarily unique—though it can be if a precise ordering of the data is desired. Multiple items can share the same sequence number as an indication that these items are to be output as a group and that the precise ordering within that group is not important or is going to be accomplished through the use of the Optimize command. Initially, unordered items share the same default sequence number of 999.

Optimizing the drill data actually rearranges the data within the drill sequence number groups—but it does not change the sequence numbers so as to maintain the desired ordering.

Items can also be marked as being 'merge data'—information which will not be panelized when virtual panelization is used, but will only be displayed on the screen and output to a drill file just once. When items are marked as merge data, their sequence numbers are displayed in a contrasting color and using Query Item will display '(Merge Data)' after the drill sequence number. See Set Order/Mark Items as Merge Data.

The Drill menu presents these following selections:

- Set Order
- Optimize
- Show Drill Path



Set Order

This command allows you to manually specify the ordering of a specified drill layer by changing drill sequence numbers of selected items. See the drill section overview above.

Drill layer

Specifies the layer that will contain selected items.

Follow Initial Selection

Specifies that selected items will be ordered after a specified item. After selecting OK in the Order Drill Items dialog you will be prompted for the item that subsequently selected items will follow—i.e. the subsequent items will have a drill sequence number of one higher than this item assigned to them.

Make Initial Selection First

Specifies that the initial selected item will be placed before all other data—i.e. this item will have a drill sequence number of one lower than the current first item in the database. This sequence number is shown.

Make Initial Selection Last

Specifies that selected items will be placed after all other data—i.e. these items will have a drill sequence number of one higher than the current last item in the database. This sequence number is shown.

Sequence Number

Specifies that the sequence number is to be manually specified for the selected items.

Item

Specifies that a selected item will be ordered.

Window

Specifies that a window will define the items to be ordered. After selecting OK in the Order Drill Items dialog you will be prompted for the points defining the window containing the desired items.

Chapter 7

Mark Items as Merge Data

When checked specifies that subsequently selected items will be marked as merge data. Merge data is output before or after all other data including step & repeat data. If unchecked, any items marked as merge data will be marked as not being merge data.



Optimize

This command allows you to optimize drill data contained in a specified drill layer. After selecting OK in the Optimize Layer dialog data contained within each drill sequence number group is optimized separately.

Optimize layer

Specifies the drill layer to optimize.

X

Specifies that the drill layer is to be optimized by performing an X sort. The quality of the X sort results are dependent on the sort data. Results are produced very quickly.

Y

Specifies that the drill layer is to be optimized by performing a Y sort. The quality of the Y sort results are dependent on the sort data. Results are produced very quickly.

Wander

Specifies that the drill layer is to be optimized by performing a wander sort. The wander sort is performed using a nearest-neighbor method that produces good results. Results are produced quickly.

Advanced

Specifies that the drill layer is to be optimized by performing an advanced sort. The advanced sort is performed using a simulated metal annealing method that produces excellent results when sufficient processing time is permitted. The time required to produce significant reductions in drilling times is very much dependent on the settings found in the Advanced Drill Optimization dialog and the CPU speed of your computer. For larger designs, using aggressive advanced sort parameters, optimizations can easily take hours to perform. See Advanced Sort Setup for more information on advanced sort parameters.

Number

Specifies that tools are to be sorted by number;

Size

Specifies that tools are to be sorted by size.

Swath Width

Specifies the swath width used when either the X or Y sorts are used.

Advanced Sort Setup

This command allows you to change parameters used when the advanced sort is performed.

Optimization Level slide control

Use this slider to select an overall optimization level. The needed parameters will be calculated based on the slider position.

Manual Override

Enabling this check button will allow you to manual set the following parameters. The absolute maximum optimization possible is usually obtainable only through a trial and error process on each job.

Starting Level

This value determines how large of a change to the original drill path, the optimization will make in one step. Values larger than 100 will sometimes do a better overall optimization of the original drill path, but will cause the optimization to take a lot longer to finish. Smaller values will cause the optimization to finish faster, but will usually not do as good of an overall job of optimization. Generally, this value should be left at 100.

Level Change Rate

This controls the number of steps the optimization will take, starting from the starting level. Larger values will improve the results of the optimization, but the results will take longer to produce. Likewise, smaller values will take less time but produce longer drill paths. Extremely low values may even cause the resulting drill file to be longer than before optimization. Values of between 0.60 and 0.95 are recommended to produce well optimized files. Values greater than 0.999 are not allowed. The default value is 0.75.

Level Duration

This controls how much time the optimization will take at each of the steps entered in the change rate above. Larger values will cause the optimization to try more possible paths at each step, while smaller values will cause fewer paths to be tried. Values of between 100 and 1000 are recommended for this variable. The default value is 100.

Chapter 7

X/Y Movement Speed Compensation

This is used to take into account the difference between the speed of the drilling machine in the X or Y axis. The default values are 1 for each, meaning the drilling head travels the same speed in each direction. If for example, the drilling head on a particular machine travels twice as fast in the X direction than in the Y, you can set the X value to 0.5 to take this into account when optimizing your files.

In general, the correct values to use for advanced optimization depend both on how good the original drill path was, and how much time you wish to spend optimizing. If the original data is in fairly good order, you can use smaller values for the level duration and level change rate. If the drill locations in the original data are at scattered locations, larger values are recommended. If at any time during your optimization, you wish to stop the program, you can press the cancel button to stop the optimization and save the best result so far.



Show Drill Path

This command allows you to highlight a drill path. The highlighting of the path can be toggled on and off using the View/Highlights command.

Drill Layer

Specifies the drill layer to highlight.

Note: After drill data has changed, this command needs to be selected again to update the displayed drill path.

Mill / Rout

The Mill/Rout command presents a sub-menu of commands as shown below:

- Create
- Edit
- Tabs
- Display

The normal steps for creating a milling file for NC milling machinery include editing the tool list (Setup/Mill Tools), creating the milling layer (Create), inserting tabs if desired (Tabs) and finally exporting the milling layer to a file (File/Export/Mill/Rout).

Create

This command allows you create milling paths and to control the way they are generated.

Mill Create Form.

Output layer

This is the layer where the milling information is to be placed. This layer must have a layer type of 'Mill'. If necessary, use the Setup/Layers command to change the layer type.

Tool

This is the number of the tool that you wish to use to complete this milling path.

Overshoot

This is an amount that can be applied to the last segment of a milling path. This can be used like a plunge line to produce a smoother edge on the milled board. Use a value of 0.0 if you do not want any overshoot.

Priority

This setting determines the order to use when processing the various mill paths. The choices range from High priority to Low priority. Items of higher priority are processed prior to ones of lower priority.

Mode settings are used to determine how the milling paths are to be created:

Automatic

This mode allows the user to select a single line and GerbTool will locate all connected lines to form the entire mill path.

Chapter 7

Interactive

This mode allows the user to select the individual lines that compose the mill path. These selected lines are then linked together and used to create the mill path.

Manual

This mode is used to create single line mill paths.

Draw

This mode is used to enable the user to draw a path that is to be turned into a milling path.

Max Outline Item Gap

When using Automatic mode, set this value to specify the maximum amount of space than can appear between items to still be considered part of the same outline.

Calculated (Compensation)

GerbTool will calculate the needed tool compensation. The milling output layer will then display the milling paths offset from the selected line segments.

Builtin

GerbTool will insert commands in the milling output to instruct the NC milling machine to apply its built-in automatic compensation. No offsets will be shown in the milling output layer.

On Line (Plunge)

GerbTool asks the user to specify a plunge point.

Add Line

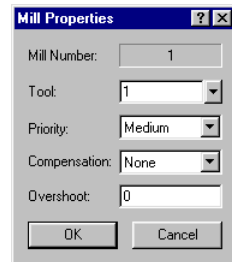
The milling tool is plunged into the board on the actual mill path.

Edit

This command presents a sub-menu of commands that can be used to edit existing milling information.

Properties

Use the Mill Properties dialog to set various characteristics of selected mill routs.



Mill Properties Dialog

Mill Number

Identifies the selected mill path.

Tool

Select the tool number you wish to use for this mill path.

Priority

Select the desired priority for this mill path. Paths of higher priority are milled before items of lower priority.

Compensation

Set the desired compensation of the this mill path. When creating paths within GerbTool with Calculated Compensation, the mill path already has been created with the tool radius compensation included.

Therefore, in these instances, this setting should be set to None so the router will not add additional compensation. When creating paths within GerbTool with Builtin Compensation, the mill path setting will be set to Right compensation. In general, you should always use right compensation for the smoothest finish.

Overshoot

Specify the amount of overshoot to be added to the end of the selected mill path if the path ends in a line segment (not an arc).

Delete Plunge

This command is used to delete the plunge line from a selected mill path or all mill paths. Once a plunge line is deleted, the milling machine will plunge directly onto the first milling line.

Chapter 7

Reverse Direction

This command is used to reverse the milling direction of already created milling paths.

Delete Mill

This command is used to delete an entire mill path.

Tabs

Use this command to insert tabs into the milling paths generated with the Mill/Create command. After specifying a tab size, simply click the mouse at the desired tab locations.

Query

This command allows you to select individual milling items and will display information about the selected mill item.

Display

This command presents a sub-menu of commands that can be used to change the way milling information is displayed.

Plunge/Retract

When enabled, displays the plunge and retract locations of the milling tool.

Offset

When enabled, displays the offset and milling directions that are to be applied to the milling path. A full arrow means no compensation is to be applied (i.e. the mill line was created with Calculated compensation), while a half arrow represents the side the offset is to be applied to. The direction of the arrow represents the direction the milling is to be performed.

Sequence

When enabled, causes sequence numbers to be displayed for each segment to be milled in each path. If due to the size of the mill segments or the current zoom level of the display, the sequence numbers would overlap, some of the sequence numbers may be omitted for clarity.

Up Path

When enabled, causes the path traced out while the mill head is retracted to be displayed as a thin line. When disabled, these movements are not displayed.

Colors

This menu item brings up a form where the user can select the colors to be used for various milling items such as plunge/retract points and the various milling priorities.

Test Points

The Test Points command presents a sub-menu as shown below:

- Tools
- Generate
- Add
- Stagger
- Save

The normal steps for creating test point data files include editing the tool list (Test Points/Tools), generating the test point locations (Test Points/Generate), editing the test locations using the Test Points/Add and Test Points/Stagger commands (as well as other normal editing commands) and finally specifying the fixture plate requirements and writing (Test Points/Save) the actual data files to disk.

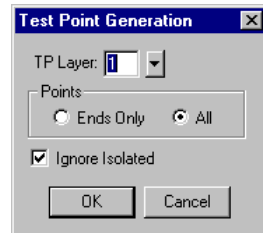
Tools

Use this command to edit the list of tools to be used when generating the test point fixture drilling files. There are fields available for editing the size and feed rate of each tool used. GerbTool will use the loaded tool list when generating the fixture plate NC files. If a tool of the exact size required is not found, the closest larger size tool will be used.

Generate

Use this command to generate the test point locations. All points considered for testing, depending on what you specify in the Test Point Generation form, will be added to the layer specified. This layer should initially be empty. Use the Setup/Layers command to create a new layer.

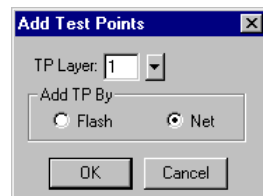
Chapter 7



Test Points Generate Form.

Add

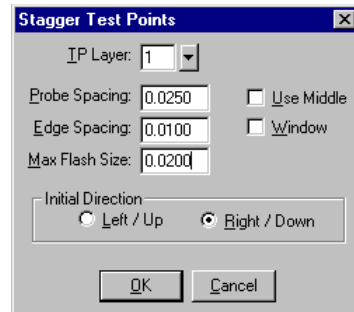
This allows easy adding of additional test points. You may add test points by flash (pad) or by net. When adding by flash, you simply click on a flash and a test point will be added to the specified test point layer with the correct netlist assignment. If adding test points by net, you click on any item (flash or draw) in a net and then click where you want a test point added. It will be included in the specified net regardless of where you place the test point.



Add Test Points Form.

Stagger

Automatic Test Point Staggering is used to provide additional spacing between test probes. As an example, fine pitch SMT components typically require Test Point Staggering to insure there is no probe to probe contact. It does this by offsetting the location of the test point from the center of the corresponding pad.



Test Points Stagger Form.

The Test Point Stagger form allows you to control the operation of the Stagger command as follows:

TP Layer

This is the layer that contains the actual Test Point locations to be probed. The active layer is assumed to be the pad master layer.

Probe Spacing

This is the minimum spacing allowed between probe points.

Edge Spacing

This is the minimum distance allowed between a staggered probe location and the edge of a pad. When a probe point is moved by the stagger function, it will not move a probe closer than this distance to the edge of a pad.

Max Flash Size

The function will not attempt to stagger probes used on pads that are larger than the specified size. If 0.0 is specified, all test points are eligible for staggering.

Use Middle

If this button is selected, the staggering function will place points in the middle of pads in addition to the edges, as long as doing so does not violate the probe spacing value. If this button is not selected, all points will be staggered to the edges of the pads.

Chapter 7

Window

If selected the user will be queried to specify a window in which the test points will be staggered. If this is not selected, the entire layer will be staggered.

Initial Direction

This allows the user to select the direction of the staggering. For example, if the setting was Left/Up, the first point that needs to be staggered is moved either to the Left or Up depending on the location of the pad which caused this point to be staggered. The next point that is to be staggered is then moved in the opposite direction and so on.

Save

This command allows you to specify all the parameters defining the probes and fixture plate requirements. GerbTool will output up to six fixture plate drill files. To enable the outputting of a fixture plate, enter appropriate values in the Filename/Dist/Thick fields. A netlist file and top plate to base fixture plate mapping file will also be generated.

	Filename	Dist	Thick
Fixture/1:	wdemo.fx	1.00	0.0500
2:	wdemo.ip2	0.50	0.0050
3:	wdemo.ip3	0.00	0.0000
4:	wdemo.ip4	0.00	0.0000
5:	wdemo.ip5	0.00	0.0000
6:	wdemo.ip6	0.00	0.0000
TP:	wdemo.tp	0.00	0.0500

Netlist File:

Mapping File:

Probe Diameter: TP Layer:

Max Deflection:

Grid Size:

Buttons: OK, Cancel, NC Format...

Save Test Points Form.

The parameters for mapping the test points to the fixture plate are configurable. The parameters include probe diameter (GerbTool adds .005 for probe to probe clearance), maximum probe deflection (this allows 1 inch and 2 inch technology etc.), and grid size (you may set this to .050 for example).

You may specify format of the output files by clicking on the NC Format button and editing the displayed form.

Convert

The convert command presents a sub-menu of commands with following selections:

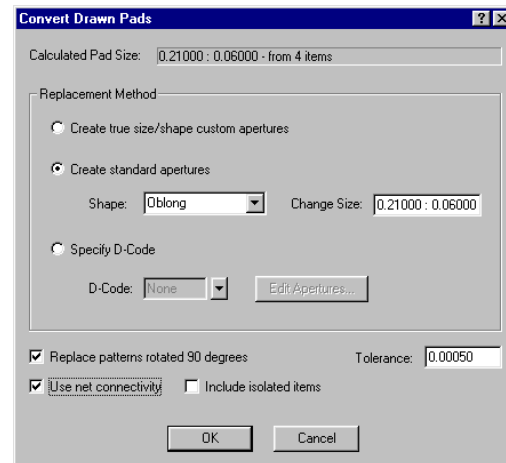
- Drawn Pads
- Home Plate
- Arcs to Segmented Arcs
- Segmented Arcs to Arcs
- Gerber to Drill
- Drill to Gerber
- To Custom
- Raster To Vector



Drawn Pads

Use this command to convert pads that are created with Gerber draws into flashes. This command should be used prior to attempting any other editing or data extraction such as Netlist generation. This command can significantly decrease the size of your database if it contains drawn pads.

After selecting an example drawn pad the following dialog box will be displayed:



Drawn Pad Replacement D-code Dialog

Calculated Pad Size

This area displays information on the selected items to be converted.

Chapter 7

Create true size/shape custom apertures

Check this radio button if you wish to have selected items converted to a single custom aperture with the same appearance.

Create standard apertures

Check this radio button if you wish to have selected items converted to one aperture of the specified shape and size.

Specify D-Code

Check this radio button if you wish to have the selected items converted to a single user-specified aperture.

Replace patterns rotated 90 degrees

Check this option if you wish to have items of the same shape rotated 90 degrees also converted.

Use net connectivity

Check this option if you wish to use netlist information used in the conversion process.

Include isolated items

Check this option if you wish to include items that are not part of a net checked for possible conversion.

Tolerance

The tolerance value allows GerbTool to increase its match frequency when the CAD system that generated the drawn pads exhibits round off errors. Usually a value of 0.0005 (inches) will suffice.

GerbTool will then locate and highlight all occurrences of any matching drawn pads and prompt you whether to continue.

Home Plate

This command is used to automatically modify pad shape for improved solder paste application using SMT stencils. After prompting for a window, the selected pad pair will be highlighted and you will be prompted whether to continue and convert all matching pad pairs.



Arcs to Segmented Arcs

This command will convert circular interpolated circles into segmented circles individually or by window. Use this command if your photoplotter can't handle circular interpolated arcs.



Segmented Arcs to Arcs

This command will convert segmented arcs to circular interpolated arcs individually or by window. Use this command to reduce the size of your database.

Segmented Arcs to Interpolated Arcs Edit Dialog.

Minimum Segments

This specifies the minimum number of segments which must be within a valid segmented arc. The higher this number, the more sure we can be that what we are looking at is an arc.

Maximum Length

Specifies the maximum length a line segment can be for it to be considered part of an arc. This prevents inclusion of traces as pieces of arcs.

Short Seg. Length

'Short Segments' can be present at the beginning and end of segmented arcs. This value explains the maximum size of a 'Short Segment', a segment which will be automatically prepended/appended to an arc. This is necessary as short segments will cause the arc-finding algorithm to fail in finding valid arcs due to insufficient precision.

Length Tolerance

Specifies the tolerance for variation of line segment lengths for segments 'inside' a segmented arc, not including the first and last segments or 'short segments'.

Chapter 7

Angle tolerance

Specifies the tolerance for the angle between a perpendicular line from the line segment and the line from the midpoint of the given line segment to the center of the arc for the purpose of discriminating as to whether the segment belongs to the arc.

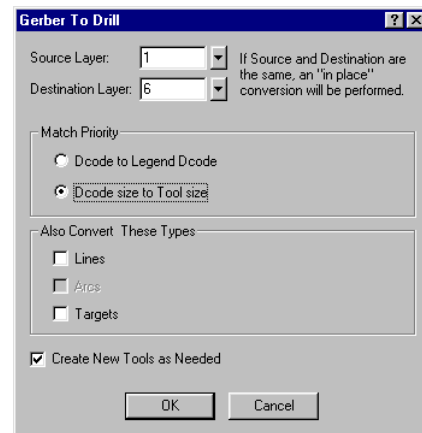
Attach Short Segs. to Lines

Specifies whether to attach 'short segments' as defined above to each other and longer segments for the purpose of eliminating these from the database.



Gerber to Drill

This command will convert Gerber data to Drill data.



Gerber to Drill edit dialog.

Match Priority

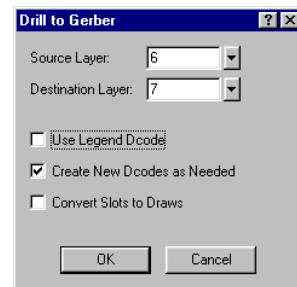
You control how GerbTool assigns drill tools during the conversion. If you specify 'Dcode to Legend Dcode', GerbTool will first attempt to match an existing dcode to the Legend Dcodes in the Drill Tool List. If a match is found, that drill tool will be assigned. If 'Dcode size to Tool size' is selected, GerbTool will attempt to match the size of the existing dcode to the size of each drill tool. If a match is found, that drill tool will be assigned. In either case, you are only assigning a priority. GerbTool will try both cases if needed to find a match.

Create New Tools as Needed

This setting, if enabled, allows GerbTool to setup new drill tools as needed if no suitable matches are found. If this is not enabled, GerbTool may not be able to convert all existing Gerber data.

Drill to Gerber

This command converts Drill data to Gerber data. Each drill 'hit' is converted to a Gerber flash and each drill slot is optionally converted to a Gerber draw.



Drill to Gerber dialog.

Source Layer

Select the Drill layer that is to be converted to Gerber data.

Destination Layer

Select the layer that is to receive the converted Gerber data. The source and target layers must not be the same.

Use Legend Dcode

Enable this check box if you want GerbTool to use the legend dcode from the tool list when converting an item to Gerber.

Create New Dcodes as Needed

This setting, if enabled, allows GerbTool to setup new dcodes as needed. If no tool legend dcode is specified or no dcode exists of the same size as the drill tool, a new dcode will be created. If this is not enabled, GerbTool may not be able to convert all existing Drill data.

Convert Slots to Draws

Enable this setting if you want GerbTool to convert Drill slots to Gerber draws.

Chapter 7

To Custom

This command converts existing database items into a custom aperture. By nature, custom apertures are graphical entities. Therefore, once converted, they do not contain any references to layer or dcode assignments and are wholly self contained. An exception to this is tool assignments. Since custom apertures are created automatically when importing drill files that contain Step and Repeat commands, tool assignments are internally stored in custom apertures. To change the tool assignments within an existing custom aperture, use the Custom Aperture Editor described in the section titled “Setup/Apertures”.

You may assign your new custom aperture to a dcode using the ‘Shape’ drop down list within the Setup/Apertures dialog.

Note: You may expand (convert) one or more instances of a custom aperture back into it’s equivalent layer data using the Edit/Dcode/Expand command described earlier in this chapter.

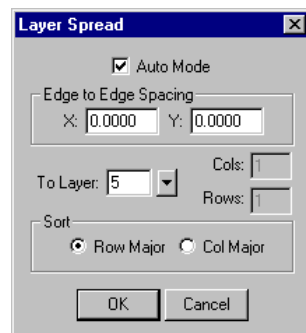
Raster to Vector

This command converts existing raster filled polygons to vector filled polygons. You specify the layer and starting dcode for the conversion. All raster polygons found on the specified layer will be converted. See the Add/Polygon command for more information on how GerbTool fills polygons with increasingly larger dcodes.



Layer Spread

Use this command to reduce your film costs by automatically copying and spreading all viewed layers onto one layer and thus one sheet of film.



Layer Spread Edit form.

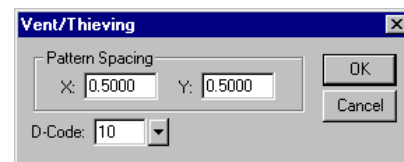
You may select automatic or manual mode using the Layer Spread edit form as shown above. If you select Auto Mode then GerbTool will automatically calculate how many images will fit in the film box as well as the position of each image. In auto mode the X&Y spacing fields specify the opposing border to border minimum spacing requirements. In manual mode, you must specify the number of rows and columns and the center to center spacing in the X&Y spacing fields. In either case, you may select either row major or column major placement. While the To Layer field may specify one of the layers to be spread, it usually is an empty layer created to accept the properly spread out images.

After clicking on the Layer Spread edit form OK button, you will be prompted to select the order in which the layers are spread. You must click on each layer to define the proper order. After doing so, the placement of all layers will be shown for your approval. If you respond affirmatively, the layers will be copied and spread as shown.



Vent/Thieving

This command allows you to manually add Venting/Thieving patterns to your database. GerbTool will display the Vent/Thieving edit dialog where you may edit the venting parameters such as pattern spacing and aperture selection.



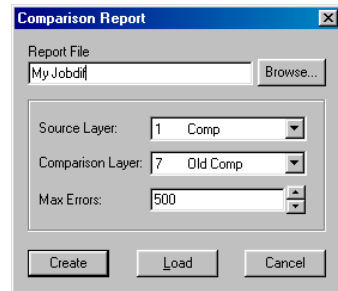
Vent Parameters dialog.

You may then define a rectangular area by entering two coordinate points. After confirmation, GerbTool will fill the specified area with a pattern of flashes as specified.

Compare Layers

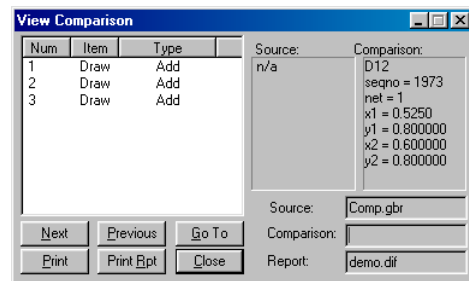
This command compares the content of two layers and graphically displays the differences. In addition to simple item by item comparisons, the output report contains the results of a netlist comparison.

Chapter 7



Compare Layers dialog.

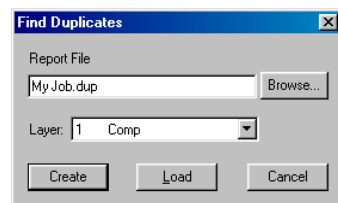
If differences are found the 'View Differences' dialog is displayed. Using this dialog you may peruse the differences and optionally make prints of the individual differences.



View Differences dialog.

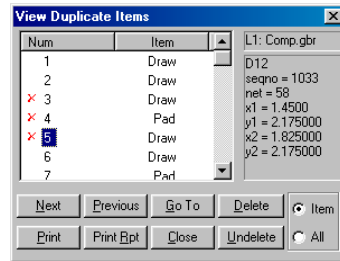
Find Duplicates

This command finds all occurrences of duplicate data. For an item to be considered a duplicate, it must be at the exact same location and have the exact same properties. A report file is generated.



Find Duplicates dialog.

If duplicate items are found, GerbTool displays the ‘View Duplicates’ dialog. Using this dialog you may peruse the duplicate items deleting them as you see fit.



View Duplicates dialog.

User Menu

The User main menu selection will display a menu of commands that are configured using the Options/Configure command. The purpose of the User menu is to allow you to make your favorite Macros and commands as accessible and easy to use as any other GerbTool command. Between the User menu and programmable mouse/function keys (see Chapter 7, ‘Options/Configure’) the commands you use the most can be truly a keystroke or mouse click away.

Chapter 8

Chapter 8

Aperture Conversion Rule Files

In addition to providing the ability to convert most popular CAD and photo-plotter aperture lists directly into the popular GerbTool format, GerbTool also allows you to create your own ACR files for specialty, proprietary or otherwise unsupported aperture list formats.

What is an ACR File

An **A**perture **C**onversion **R**ule file is a simple ASCII file used to describe a particular aperture list format using easy to use conversion language statements. Using your favorite text editor, you describe the expected format of the aperture list. Once complete, GerbTool will then be able to convert this new aperture list format automatically just like any other 'supported' aperture list format.

How does it work?

GerbTool first reads in an ACR file. This file describes the format of the original aperture list, and how it is to be converted into a GerbTool aperture file. Once the ACR file has been read in, GerbTool then converts the aperture lists in question.

Regular Expression Syntax

GerbTool ACR files use something called a regular expression syntax to describe lines that are to be processed by the aperture list converter. A regular expression is simply a way of describing a certain pattern of text in a well defined fashion. The expression syntax ACR uses has many sophisticated features, many of which you will probably never need. They are included however, in case they are needed.

What follows is a description of the regular expression syntax GerbTool ACR files use. We will first start out with the simplest types of patterns that are supported, and then progress to the more advanced ones. The simplest expression is the character. A character in the rule file will match the corresponding character in the aperture list. For example the rule:

`D$dcode`

will match the line

`D10`

in an aperture list, but not the line

`DCODE10`

due to the extra characters 'CODE'. Some characters have special meaning in a rule. These characters include '.', '+', '*', '(', ')', '[', ']', '?', '\$', '^', '\'. It is unlikely that you will encounter any of these characters when creating an aperture converter of your own, but if you do need to use one of them, you can precede the character with a backslash '\'. For instance the string:

```
\ (D$dcode\)
```

matches

```
(D10)
```

Quite often you will have a variable number of characters to match. This occurs most often with spaces. Patterns of this sort can be matched by using the characters '+' and '*'. A character followed by '*' matches 0 or more instances of that character. A character followed by '+' matches 1 or more instances of that character. For example, the string:

```
ab+a
```

matches all of the following:

```
aba  
abba  
abbba  
abbbba
```

while the string:

```
ab*
```

matches all of those lines plus the line:

```
aa
```

Chapter 8

Matching Order

Sometimes you will encounter instances where a line in an aperture list will match multiple FORMAT lines in your aperture list converter. If this happens, the converter will use the first FORMAT line, matched. For example, to match the following 2 lines:

```
10 THERMAL 0.25 0.15          (normal thermal)
11 THERMAL 0.25 0.15 45.0     (45 degree thermal)
```

You would want to use the following 2 lines in this exact order:

```
FORMAT_THERM45:$dcode THERMAL +$xsize +$ysize +45.0      (a)
FORMAT_THERMAL:$dcode THERMAL +$xsize +$ysize             (b)
```

If you were to have the order of these lines reversed, line (b) would match both of the original aperture list lines, and the converter would produce two non rotated thermals.

Creating an ACR File

An ACR file contains two types of statements. The first type of statement describes the environment such as the expected file extension, metric mode, number of header lines to skip etc.. The second type of statement is the actual rule statement. Rules are the statements that will be used to match incoming aperture list entries to a corresponding GerbTool aperture shape. The following is a description of each type '1' ACR statement and the expected parameters if any:

NAME

Syntax: NAME converter_name

Parameters: converter_name
The name of the ACR file. Should be a single word.

Description: This statement will place the parameter in the header of the resulting aperture list.

Example: The following example sets the name of the converter to 'Allegro'.

```
NAME Allegro
```

VERSION

Syntax: VERSION version_number

Parameters: version_number
 The version number of the ACR file. The version number should be a single decimal number.

Description: This statement will place the parameter in the header of the resulting aperture list.

Example: The following example sets the version number of the converter to '6'.

```
VERSION 6
```

HEADER

Syntax: HEADER lines_to_skip

Parameters: lines_to_skip
 The number of lines to skip in the header of the aperture list.

Description: If this line is present, the number of lines specified will be skipped from the header of the aperture list file you are attempting to convert. This can be used to quickly bypass information at the top of a file that you know does not contain any apertures.

Example: The following example instructs GerbTool to skip the first twenty lines of the aperture list.

```
HEADER 20
```

SKIP

Syntax: SKIP skip_string

Parameters: skip_string
 A text string to mark text to be skipped.

Description: If this line is present, all lines in the aperture list that start with the given character string will be ignored.

Chapter 8

Example: The following example will allow GerbTool to skip over lines that begin with 'MOIRE'.

```
SKIP MOIRE
```

DEFAULT_UNITS

Syntax: DEFAULT_UNITS mode

Parameters: mode
One of "\$INCH", "\$MIL" or "\$MM"

Description: If given, will cause the values read in to be interpreted as Inches, Mill's or Millimeters, depending on the value used.

Example: The following example sets the units mode to metric.

```
DEFAULT_UNITS $MM
```

CUSTOM

Syntax: CUSTOM yesno

Parameters: yesno
Either "\$YES" or "\$NO".

Description: If set to \$YES, GerbTool will attempt to create custom aperture names whenever possible. Otherwise a Diamond shape will be substituted. NOTE: GerbTool will not create the custom apertures themselves, only their names in the aperture list.

Example: The following example sets the creation of custom apertures to off.

```
CUSTOM $NO
```

EXTENSION

Syntax: EXTENSION extension

Aperture Conversion Rule Files

Parameters: extension
The default aperture list extension.

Description: The default extension of the aperture lists you will be converting with this rule file. If the value is entered here, you will not need to enter it when specifying the aperture list for conversion.

Example: The following example sets default aperture list extension of 'mya'.

```
EXTENSION mya
```

DEBUG

Syntax: DEBUG mode

Parameters: mode
A value of 0, 1 or 2.

Description: Enables debugging information to be output into the aperture converters log file. If the value of 0 is used, no debug information will be output. If 1 is used, GerbTool will output debug information while parsing the ACR file, and if the value is set to 2, debug information will be output while converting the aperture file itself. This line is for advanced users only and should either not be included or be set to 0 for normal converter operation.

Example: The following example sets the current debug mode to 2.

```
DEBUG 2
```

XTENSION

Syntax: XTENSION dll_filename

Parameters: dll_filename
The name of a DLL that you supply.

Description: If present causes the converter to look for the specified 'DLL' file to help in converting the aperture lists.

Chapter 8

Please contact WISE Software Technical Support Services for more information on developing DLL's to assist in converting 'difficult' aperture list formats.

Example: The following example specifies a user supplied DLL.

```
XTENSION myapfmt.dll
```

DCODE

Syntax: DCODE mode

Parameters: mode
One of "\$\$ONLINE", "\$\$SEQUENTIAL" or "\$\$GERBER_ORDER".

Description: This line controls how D-code values will be derived. If set to \$\$ONLINE (the default) the codes read on each line will be used. If \$\$SEQUENTIAL is used, lines that match the rules given will be assigned sequential numbers. Some aperture lists have their D-codes arranged in a special non-sequential order used in certain Gerber photoplotters. This order will be used if \$\$GERBER_ORDER is set.

Example: The following example sets the D-Code mode to sequential.

```
DCODE $$SEQUENTIAL
```

#

Syntax: # any_text

Parameters: any_text
The body of a comment.

Description: This symbol leads comments in an ACR file.

Example: The following example shows a typical comment.

```
# Created By Joe Designer
```

The following is a description of each rule type of ACR statement and the expected parameters if any:

FORMAT_shape

Syntax: FORMAT_shape rule

Parameters: shape

The possible shapes are: ROUND, SQUARE, RECT, OBLONG, DONUT, DIAMOND, OCTAGON, THERMAL, THERM45, TARGET, CUSTOM. Please note that this parameter should be combined with the 'FORMAT_' statement to form a single word such as 'FORMAT_ROUND'.

rule

A rule for matching apertures that are to be mapped to a GerbTool 'shape' aperture.

Description: If the rule matches a line in the aperture list being converted, that line will be converted into a GerbTool 'shape' aperture.

Example: The following example will match the line "JUNK D10 0.060 0.060 ROUND".

```
FORMAT_ROUND $skip +D$dcode $xsize $ysize  
ROUND
```

FORMAT_UNITS

Syntax: FORMAT_UNITS rule

Parameters: rule

A rule for matching a line in the aperture list that specifies the format of the file.

Description: A line matching this is used to determine the format of the aperture list. This statement allows the aperture list itself to override a previous UNITS statement.

Chapter 8

Example: The following example will match the line “FORMAT MM”.

```
FORMAT_UNITS $skip $units
```

FORMAT_SPECIAL

Syntax: FORMAT_SPECIAL rule

Parameters: rule
A rule for matching lines for use by an XTENSION DLL.

Description: Does not produce a GerbTool D-Code line. It is used for special processing by an XTENSION specified DLL.

Example: The following example will match the line “SQR D10 0.060 0.060”.

```
FORMAT_SPECIAL SQR +D$dcode $xsize $ysize
```

When constructing rules to match apertures, there are special key words that you place in the rule that will cause GerbTool to assign the values contained in the desired fields to the corresponding GerbTool aperture list fields. These keywords are as follows:

Keyword	Meaning
\$dcode	Assigned to D-Code
\$xsize	Assigned to xsize
\$od	Assigned to xsize
\$ysize	Assigned to ysize
\$id	Assigned to ysize
\$rot	Assigned to rotation
\$tool	Assigned to tool num
\$skip	Skip this field
\$custom	Use this field to make a custom aperture
\$units	Used to determine the format of the aperture list.

Aperture Conversion Rule Files

The following is a sample ACR file:

```
# Aperture converter for Mentor

NAME Mentor
VERSION 1.0
EXTENSION rpt

# handle swapped X/Y columns
XTENSION mentor.dll

DEBUG 0

CUSTOM $$NO

DEFAULT_UNITS $$INCH

HEADER 1

FORMAT_ROUND $skip +circle +$skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_THERMAL $skip +circle +$skip +$xsize +$ysize +$rot +false +true +$dcode
FORMAT_RECT $skip +rectangle +$skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_SPECIAL Position +Shape

# Mentor now has multiple formats
FORMAT_ROUND +$skip +$dcode +circle +$skip +$xsize +$ysize
FORMAT_THERMAL +$skip +$dcode +circle +$skip +power +$xsize +$ysize
FORMAT_RECT +$skip +$dcode +rectangle +$skip +$xsize +$ysize
FORMAT_SPECIAL Aperture Position
```

Chapter 9

274-X

Chapter 9

274-X

GerbTool supports the versatile “eXtended” Gerber data format, 274-X, developed by Gerber Systems, Inc. (GSI). This format provides for the inclusion of aperture data directly in the Gerber data files (embedded apertures), flexible aperture definitions and easy single file compositing.

Embedded Apertures

Note: While it not necessary to understand the syntax of 274-X to manipulate 274-X files within GerbTool, several examples of 274-X syntax are provided below. These examples are provided to acquaint you the 274-X only. Please refer to the instruction manuals provided with your photo plotter or contact GSI directly for more information on the 274-X syntax.

A 274-X format Gerber file contains all aperture definitions necessary to plot the data thereby eliminating the need for an external aperture list. An aperture is defined within a 274-X file with an AD command as follows:

```
%ADD<code><macro_name>,<parameter_list>*%
```

example:

```
%ADD10C,0.06X0.020%
```

This example defines D10 as a simple 60 mil round flash using the GSI intrinsic aperture macro 'C'.

Aperture Macros

Aperture Macros are used to describe the size and shape of special apertures. Using aperture macro primitives, it is possible to design very complex aperture shapes. Each primitive describes a basic shape such as a circle or a line. Each primitive also specifies its polarity (on/off) allowing data to be removed for such features as donuts or spokes in a thermal. Shown below are the different primitives available.

274-X

#	Type	Parameters...
1	Circle	on/off diameter xcenter ycenter
20	Line-Vector	on/off width xbeg ybeg xend xend rot
21	Line-Center	on/off width height xcenter ycenter rot
22	Line-Lowerleft	on/off width height xloc yloc rot
4	Outline	on/off count x y... rotation
5	Polygon	on/off sides xcenter ycenter diameter rot

274-X Aperture Macro Primitives.

Aperture macros are also programmable by using replaceable parameters. Replaceable parameters allow a macro to produce different results depending on the aperture definition specified by the AD aperture definition command explained in the preceding section. Replaceable parameters are indicated by a dollar sign ('\$') followed by a numeric value. The numeric value indicates the parameters position within the AD aperture definition. A typical donut macro and corresponding definitions are shown below:

```
%AMDONUT*
1,1,$1,0.0,0.0*
1,0,$2,0.0,0.0*
%

%ADD10DONUT,0.60X0.40%
%ADD20DONUT,0.08X0.70%
```

In the above example, D10 is defined as a 60 mil donut with a 40 mil hole and D20 is defined as a 80 mil donut with a 70 mil hole. Note that both D10 and D20 refer to the same macro but have different sizes!

Layer Compositing

274-X allows a single Gerber file to define a composite image of arbitrary complexity. Each "layer" of data within the Gerber file is simply prefixed with an appropriate polarity command. Ordering of the layers is critical as the data is processed sequentially. For assistance, check the example files provided and notice how each layer either adds or removes from the initial image.

GerbTool automatically creates separate layers for composite layers when reading a 274-X file and conversely creates a single file for all layers that form a composite when writing out data.

Viewing Composites

Composite layers may be displayed WYSIWYG by simply typing the ‘V’ nested command. This nested command toggles composite viewing on/off. When **enabled**, composite layers will be displayed as they will plot. When **disabled**, composite layers will be displayed as if all layers were dark (positive). Composite viewing may also be controlled using the Setup/Composites form.

Converting From 274-D To 274-X

In order to convert a set of standard Gerber 274-D files into a single Gerber 274-X composite file, load the 274-D files as you normally do and then perform the following steps using the Setup/Layers and Setup/Composites commands:

- ✓ Set the Layer Name field of each layer to a meaningful name.

Hint: Setting the Layer Name field to the original filename of the of the same layer will label the 274-X “layers” in a fashion that will be familiar to the user.

- ✓ Use the Setup/Composites command to create a new composite, assign layers and set the desired polarity of each layer. A polarity of Dark means that the layer is to be displayed in the style a normal Gerber file is displayed. Clear tells GerbTool to display the layer using the current background color. This has the effect of erasing, or “clearing”, areas from an image that were previously drawn by a “dark” layer. Negative layers should be set to clear.
- ✓ Save the composite file using the File/Export/Gerber command.
- ✓ Decide what filename you wish to use for the new Gerber 274-X file and rename all of the Gerber filenames to this new name. It is important that each 274-X “layer” have the same filename.
- ✓ Click on the Format button. Change the Dialect field to 274-X.

To load this new 274-X composite file into another design, enter its filename as you would with any other Gerber file, making sure the file format has been set to 274-X. There is no need to load in an aperture list as it is included in the 274-X file.

Chapter 10

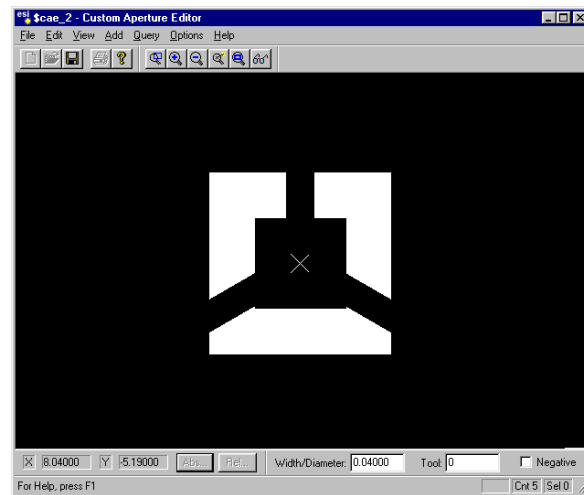
Using Custom Apertures

GerbTool allows you to create custom apertures quickly and easily. A custom aperture is a collection of graphic drawing primitives such as circles and lines. This Chapter details the steps for creating a custom aperture.

Chapter 10

Create Custom Aperture

The easiest method of creating a custom aperture is to use the Tools/Convert/To Custom command. This command allows you to window around a group of items on a layer and have GerbTool automatically convert the selected data into a custom aperture. Once you have a custom aperture created you may use the Setup/Apertures command to invoke the Custom Aperture Editor. This tool allows you to make changes to your custom aperture using a graphical editor as shown below.



Custom Aperture Editor.

Chapter 11

Working with Text Fonts

GerbTool uses a font file containing a list of X-Y coordinate pairs that constitute the ‘strokes’ required to display each character inserted by the Add/Text command. You may have more than one font file but GerbTool will always read the file ‘stroke.fnt’ at startup. To use a different font file, simply rename the current ‘stroke.fnt’ to some other meaningful name and then rename your font file to ‘stroke.fnt’. GerbTool allows you to edit existing fonts and create new fonts that are to be used for text insertion. This Chapter details the steps for editing fonts.

Editing a Font

Before you edit a font you must convert it into individual Gerber files for each character. To do this, from the system prompt change to the GerbTool 'fonts' directory and type the following command:

```
f2g ../stroke.fnt [Enter]
```

This will create an individual Gerber file for each character in the font file. You may now start GerbTool and load one of the provided design files upcase.dsn, lwcase.dsn, numbers.dsn, punc1.dsn or punc2.dsn which cover uppercase, lowercase, numbers and punctuation characters respectively. The Film Box is set to a 7mil square which each character must remain within. You may draw any shape you desire as long as you stay in or on the Film Box and you don't try to add flashes.

Note: It is important that the file format of the individual Gerber files for each character remain at Imperial, absolute, '2.3' and no zero suppression.

Once you have finished editing the characters you wish to modify you may use the following command, at the system prompt, to create a new font file:

```
g2f newfile.fnt [Enter]
```

In the above example a new font file would be created with a file name of 'newfile.fnt'. Please note that this program does not purge the individual Gerber character files. You may do this manually if desired. Remember, that GerbTool will not recognize your new font file unless it is named 'stroke.fnt' and is in the GerbTool program directory.

Creating a New Font

To create a completely new font you may follow the steps detailed in 'Editing a Font' above, but skip the font file to Gerber file conversion step.

Note: It is usually easier (and faster) to modify an existing font than to create one from scratch.

APPENDIX A

Aperture List File Format

This Appendix describes the format of a GerbTool aperture list and provides an example of a actual aperture list.

APPENDIX A

Aperture lists are stored as simple ASCII files. There are nine fields in each line of the file. Each line defines one D-Code. The fields consist of the following:

Field	Possible Values	
D-Code	10 - 9990	
Shape	Round, Rectangle, Square, Oblong, Donut Diamond, Octagon, Thermal, Thermal45, Target, or a V8 custom aperture filename prefixed by a '%'	
Width	0.0 - 9.9999	
Height	0.0 - 9.9999	When referring to Donuts or Thermals, this field represents the diameter of the inner hole. When referring to Targets, this field refers to the diameter of the inner ring of the Target.
Type	SMT, Thruhole or Thermal	
Tool	0-999	Not used after Version 8.
Tool Size	0.0 - 9.9999	Not used after Version 8.
Legend	10 - 4095	Not used after Version 8.
R90	10 - 4095	Specifies the D-Code to substitute for this D-Code when rotating 90 or 270 degrees. This field exists only for compatibility with older versions of GerbTool as newer versions perform the D-Code substitutions automatically.

Aperture List field definitions.

All fields are separated by white space. Lines that begin with a '#' are treated as comments. Although the author and data comments are not required, they are generally included as an aid for other users. The header of a GerbTool aperture list may contain a format line proceeded by a '%'. This line contains either 'IMPERIAL' or 'METRIC' followed by a version number. If IMPERIAL is specified, all sizes are in inches. If METRIC is specified, they are in millimeters. If no format line is provided, IMPERIAL is assumed. The version number is for

Aperture List File Format

documentation purposes only. An excerpt from an aperture list showing the required format follows:

#	Format, Version								
#	%IMPERIAL, V3.0								
#									
#	Author: GerbTool V1.0 (c) 1992 WISE Software Solutions, Inc.								
#	Date: Wed Oct 7 13:28:46 1992								
#									
#	Shape	Width	Height	Type	Tool	Size	Legend	R90	
#									
D12	Round	0.0100	0.0100	THRU	0	0.0	0	0	
D21	Square	0.0200	0.0200	THRU	2	0.0	0	0	
D22	Rectangle	0.0220	0.0180	SMT	3	0.0	85	0	
D23	Oblong	0.0220	0.0180	THRU	3	0.0	0	0	
D24	Diamond	0.0240	0.0240	THRU	4	0.0	0	0	
D25	Target	0.1800	0.1600	THRU	0	0.0	0	0	
D26	%FIDUCIAL	0.0000	0.0000	THRU	0	0.0	0	0	
D70	Octagon	0.0240	0.0240	THRU	5	0.0	0	0	
D71	Thermal	0.0240	0.0200	THRU	0	0.0	0	0	

Sample Aperture List file.

In the above example, D26 is specified as a Version 8 style Custom aperture with a filename of 'fiducial.cus'. The '%' is required so GerbTool knows that what follows is a custom aperture filename.

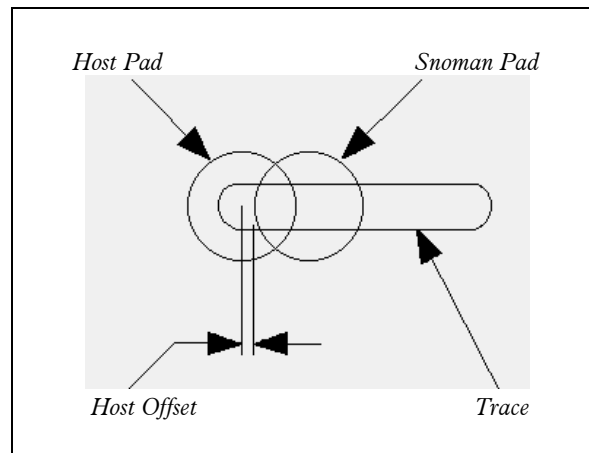
APPENDIX B

APPENDIX B

Snoman Concepts

Snoman is a tool designed to create a *maximum material condition* at the point where a trace segment enters a pad, thereby eliminating the possibility of pad/trace separation (breakout). This is accomplished by examining a Gerber file (layer) and outputting pad flashes at the correct locations, and of the correct size, to provide the most material where a trace enters a pad. Automatic adjustments are made to the size and location of the generated *snoman* pads to eliminate design rule spacing violations.

The following illustration shows the original pad and trace as well as the resultant *snoman* pad.



Snoman Concept.

The distance maintained between the host pad center and the edge of the generated *snoman* pad (see Host Offset in illustration above) is adjustable. Negative values allow the *snoman* pads to closely hug the host pads if desired.

APPENDIX C

APPENDIX C

Sample Netlist File

The following is a sample netlist file created by the
Tools/Netlist/Generate and Tools/Test Points/Save commands:

```
NET 1
 825,4600 2300,5225 2300,5325 2700,4750
NET 2
 875,4700 975,4700
NET 3
 900,4500 1300,1600 1300,4200 1350,4500
NET 4
 900,4600 2400,5175 2400,5275 2400,5375
NET 5
 975,3375 1700,3375 1700,3475
NET 6
 975,4000 1100,3900 1600,4200
NET 7
 3300,4000 3400,3375 3400,3700 3400,3800 3400,3900
3500,3675 3500,4050
NET 8
 975,4800 1300,5000
NET 9
 1000,4400 1000,4500 1100,4200 1200,1600
NET 10
 1100,3375 1500,2725 2125,2625
NET 11
 1200,2325 1400,1000 1400,2725 2725,750
NET 12
 1200,3075 2275,3050 2275,3150
NET 13
 1300,3375 2125,2525
NET 14
 1400,1300 1600,1300 1600,2725 2525,750
```

Sample Netlist file.

APPENDIX D

APPENDIX D

Color List File

When starting up, GerbTool loads a color list file, named *color.rgb*. GerbTool reads first the available colors from a RGB color/name pair list and second, a list of current choice colors. The current choice colors specified will be those presented whenever you select colors within GerbTool (i.e. flash and draw colors). You may modify both the color/name pair list and the color choice list to meet you needs.

```
# maximum 1024 colors available...
[RGB Color/Name pairs]
128  0  0          vga16red
  0 128 128        vga16cyan
  0 128  0          vga16green
245 245 245        WhiteSmoke
.
.
.
.
255 250 240        FloralWhite
253 245 230        OldLace
250 240 230        linen
250 235 215        AntiqueWhite

# maximum 32 current choice colors...
[Choice Colors]
blue
vga16green
white
black
coral
.
.
.
.
SteelBlue
SaddleBrown
DarkSalmon
DarkOrange
DeepPink
```

Sample Color List.

Glossary

Glossary

Absolute mode

When all X-Y coordinates are referenced to a common origin (0,0).

Active Layer

The layer that all items added to the database will go.

Aperture List

A list of Gerber D-Code definitions.

ASCII

Acronym for American Standard Code for Information Interchange.
This is a standard that relates characters to specific code numbers.

Block size

The size of a coordinate value in characters. Also known as m.n format.

Breakout

Pad and trace separation during manufacturing.

Clicking

Pressing and releasing a mouse button.

Desktop

The screen background for GerbTool on which Gerber data, menus, icons and dialog boxes appear.

Glossary

Design File

A file containing information about the layer structure of a single PCB design. This file also stores various information about the GerbTool operating environment.

Double Clicking

Pressing a mouse button twice in rapid succession.

DRC

Acronym for Design Rule Check.

Incremental mode

When each X-Y coordinate is a displacement from the previous coordinate.

Isolated Pads

Pads that do not have a trace connected to them.

Mouse Cursor

An icon that indicates the current mouse position.

NC Drill

Within GerbTool this refers to files produced to drive Numerically Controlled drilling machines.

Netlist

A file containing groups of pad X-Y locations that are connected by traces.

Pad Removal

The act of removing isolated or stacked/buried pads.

Pan

Moving the location of the viewing window without changing its size.

Panelize

Placing multiple copies of a PCB on one piece of film. The multiple copies are then manufactured on a single panel thereby reducing manufacturing costs.

Point

A X-Y location within the Drawing Area.

Scroll Bar

A box within a form used to scroll the contents of the form. Move the mouse over the box and press the left mouse button. Without releasing it, move the box up or down by moving the mouse. When you release the mouse button the form will scroll.

Virtual Memory

A combination of hardware and software that allows an application to address all memory that the CPU is capable of addressing even when there is less actual memory. The virtual memory manager swaps data back and forth to the disk and remaps memory addresses to provide applications with virtually unlimited memory. Available disk space becomes the limiting factor.

Glossary

Index

2

274-X, 40, 48, 169

A

absolute mode, 187
ACR files, 157
active layer, 15
add, arc, 73
add, arc 3 pt, 74
add, array, 74
add, circle, 73
add, draw, 73
add, drill, 73
add, flash, 72
add, polygon, 74
add, rectangle, 73
add, slot, 73
add, text, 75
add, vertex, 73
aligning layers, 28, 66
aligning pads, 66
annular ring, 126
aperture list, 118
aperture list, convert, 157
aperture list, embedded, 169
aperture list, format, 178
aperture list, setup, 78
aperture list, use report, 86
apertures, custom, 173
apertures, macro, 170
apertures, maximum size, 106
arcs, chord angle, 107
arcs, convert to interpolated, 149
arcs, convert to segmented, 148
arcs, interpolated, 15, 104
ASCII, 187
Auto correction, DRC, 128
Auto correction, Silkscreen, 132

B

background color, 109
BARCO DPF, 42, 51
basics, 11
blind&buried vias, 84, 121

block size, 187
breakout, 181, 187

C

CAM tools, 118
chamfer, 63
chord angle, 107
clip data, 63
color chooser, 20
color list file, 185
color, background, 109
command reference, 35
composites, 274-X, 171
composites, setup, 82
composites, viewing 274-X, 70, 172
composites, viewing/plotting, 30
configure, aperture converters, 111
configure, display, 108
configure, function keys, 110
configure, general, 105
configure, macro files, 113
configure, mouse, 110
configure, paths, 112
configure, right click menu, 116
configure, units & precision, 115
configure, user menu, 114
conversion, 274-D to 274-X, 172
conversion, aperture lists, 157
conversion, drawn pads, 147
conversion, Drill to Gerber, 151
conversion, Gerber to Drill, 150
conversion, home plate, 148
conversion, raster to vector, 152
conversion, to custom, 152
coordinate display, 16
coordinates, absolute, 19
coordinates, relative, 19
copper area calculation, 102
copy, 62
crosshair, size, 109
custom apertures, 173
custom apertures, convert to, 152
custom fonts, 175

Index

D

data extents calculation, 103
d-code, expand, 65
d-code, highlight, 101
d-code, polarity, 65
d-code, scale, 65
d-code, transcode, 32, 65
delete, 63
design file, 188
design file, merge, 36
design file, new, 8, 36
design file, open, 8, 36
design file, save, 36
design file, save as, 37
desktop, 12, 187
destination layer, 62, 94
dialog boxes, 20
dimensioning, auto, 92
display units & precision, 15
display, units & precision, 105
documentation tools, 86
documentation, redline/markup, 89
double clicking, 188
drawing area, 13, 16
drawing cursor, 17
drawing, construction lines, 95
drawing, dimensioning, 92
drawing, fabrication, 91
drawing, hole charts, 96
drawn pads, conversion, 30, 147
DRC, 124, 188
DRC, annular ring, 126
DRC, Auto correction, 128
DRC, keepouts, 127
DRC, missing drill, 126
DRC, stubs, 126
DRC, view errors, 127
DRC, well behaved, 126
drill tools, change, 66
drill tools, setup, 80
drill tools, use report, 87, 88
Drill, export, 29
Drill, import, 29, 42
drill, optimize, 136
drill, set order, 135

drill, show path, 138
DXF, export, 52
DXF, import, 45

E

edit item, 61
email address, 4
email, design, 58
exiting, 9
exiting GerbTool, 58
export, Aperture List, 50
export, BARCO DPF, 51
export, bitmap, 55
export, drill, 50
export, DXF, 52
export, Gerber, 47
export, HPGL, 52
export, IPC-D-350, 51
export, Mill/Rout, 56
export, PostScript, 54
export, Tool List, 51
export, V8 design, 56
extents, 103

F

file chooser, 21
fillet, 63
film box, 17
film box, change, 108
find duplicate data, 154
FIRE9XXX, 40, 48
floating pad removal, 133
fonts, creating, 176
fonts, custom, 175
fonts, editing, 176
fonts, TrueType, 76
format, char set, 41, 49
format, critical, 41, 42
format, mode, 41, 49
format, zero suppression, 41, 49
function key, programming, 17

G

Glossary, 187

grid, change size, 108
grid, display, 15, 70
grid, snap, 15, 104
group, selecting, 60

H

help button, 20
highlight, clear, 70
highlight, colors, 109
highlight, d-codes, 101
highlight, nets, 99
highlight, on/off, 70
highlight, selections, 70
highlight, userdata, 100
home plate conversion, 148
HPGL, 43, 52
HPGL, interactive, 53

I

import, Aperture List, 41
import, BARCO DPF, 42
import, CAM files, 8
import, Drill, 42
import, DXF, 45
import, Gerber, 40
import, Gerber Wizard, 37
import, HPGL, 43
import, IPC-D-356, 43
import, Mill/Rout, 47
import, Tool List, 42
incremental mode, 188
installation, 5
introduction, 1
invoking commands, 17
IPC-D-350, 51
IPC-D-356, 43
isolated pads, 188
item info, displaying, 98
item, editing, 61

J

join, 63

L

layer sets, setup, 83
layers, active, 15
layers, color&visibility, 12
layers, compare, 153
layers, maximum, 106
layers, rearranging, 78
layers, ref mode, 107
layers, registration, 28, 66
layers, setup, 77
layers, spread, 152
license request, 6

M

Macro Developer, 117
macro, edit, 117
macro, load, 117
macro, record, 118
macro, run, 117
macros, function keys, 110
markup, 89
measure, center to center, 101
measure, edge to edge, 101
measure, point to point, 101
memory considerations, 25
memory, allocation errors, 25
memory, compact, 67
mill, colors, 143
mill, create, 139
mill, display, 142
mill, edit, 140
mill, query, 142
mill, tabs, 142
Mill/Rout, 47, 138
Mill/Rout, import, 47
mill tools, setup, 82
mirror, 64
mouse, programming, 17
move, 62

N

NC Drill, 188
nested commands, 18, 24, 116
netlist, 188

Index

netlist, enable/disable, 49
netlist, generate, 121
netlist, highlight, 99
netlist, sample, 183
netlist, saving, 123
netlist, well behaved, 122

O

offsets, applying, 64
options, 104
origin, changing, 67
ortho, line angle, 107
orthogonal mode, 15, 104
overlay mode, 14
overview, 7

P

pad removal, 188
pad removal, isolated, 133
pad removal, stacked, 134
pads, registration, 66
page setup, 57
panelize, 119, 189
panelize, automatic, 29, 119
panelize, manual, 119
panelize, virtual, 50, 120
panning, 68, 189
performance tips, 23
plotting, batch mode, 53, 55
plotting, borders, 53, 55
plotting, composite, 54
polygon, filling, 74
polygon, pouring, 74
PostScript, 54
print preview, 57
printer setup, 58
printing, 57
programming, function keys, 25
programming, mouse, 25
purge, 67

Q

query, item, 98
query, nets, 99

query, userdata, 100
quick start, 7

R

redlining, 89
redraw, interrupting, 20, 24
redraw, minimize, 109
registering layers, 28
report, aperture list, 86
report, drill tools, 87, 88
rotate, 64
rule violation, displaying, 14, 71

S

sample net list file, 183
save, 36
save as, 37
save, modified data, 9
scale, database, 64
scale, D-Codes, 65
scroll bars, 189
selecting groups, 60
selection filter, 58
selection filter, turn off, 14
settings, bar, 13
setup, install program, 5
silkscreen, auto fixup, 31, 132
sketch mode, 14
Snoman, 33, 130
Snoman Concepts, 181
soldermask, create, 32
speeding up commands, 24
split screen, 13
starting GerbTool, 8
status bar, 17
step & repeat, 50, 120
stick mode, 109
surface mount, 79
system requirements, 3

T

Teardrops, 131
Teardrops, T-Junctions, 132
technical assistance, 4

terminating a command, 20
test points, add, 144
test points, generate, 143
test points, save, 146
test points, stagger, 144
test points, tools, 143
text, 75
thieving, 74, 153
thru-hole, 79
T-Junctions, Teardrops, 132
tool tips, 13
toolbar, 12
transcoding, 32
transparent viewing, 14

U

Undo, 24, 59
undo, on/off, 107
unused pad removal, 133
User Menu, 155
userdata, editing, 61
userdata, highlight, 100
userdata, saving, 49

V

vent, automatic, 120
vent, manual, 153
view, all, 69
view, birdseye, 16
view, composites, 14, 70
view, end cap, 108
view, errors, 14, 71, 127
view, film box, 69
view, grid, 70
view, overlay mode, 69
view, pan, 68
view, previous, 72
view, recall, 71
view, redraw, 69
view, save, 71
view, sketch mode, 69
view, split, 72
view, split screen, 16
view, toolbars, 72

view, virtual panel, 70
view, window, 68
view, zoom in, 68
view, zoom limit, 108
view, zoom out, 68
virtual memory, 25, 189

W

web site, 4
well behaved, DRC, 126
well behaved, netlist, 122
wizard, import, 106